 ISSN : 2249-5762 (Online) | ISSN : 2249-5770 (Print)

Analysis of External Aerodynamics of Sedan and Hatch Back Car Models Having Same Frontal Area by Flow Visualization Method

[1]Sharath Kumar S N, [2]Dr. Sathish S
[1,2]Dept. of Mechanical Engineering, Dr. Ambedkar Institute of Technology, Bangalore, India
[1]Honorary Consultant, Rigel Research and Engineering Solutions, Bangalore, India

Abstract
Aerodynamicist have a difficult task to perform, as they deal with invisible medium i.e. air, for their aerodynamic analysis. Greater understanding and valuable insights of the physical behavior of an air flow such as boundary layer formation, flow separation and vortex generation etc., can be known if the entire flow field was made visible by some technique which could be seen by our naked eyes or by a recording device. One such method is the flow visualization technique. In the present study aerodynamic analysis by flow visualization technique is carried out for Sedan and Hatch back car models to checkout their aerodynamic efficiency. Flow over both Sedan and Hatch back car models is found to be same up to roof and rear wind shield intersection, after which flow separation occurs in both the car models. Due to presence of boot in case of sedan car model there is a slight flow reattachment. The vortex formation at the rear in case of Hatch back car model is more as compared to Sedan Car model. Hence it can be concluded that Sedan Car model is more steam lined, aerodynamic and experiences less drag force as compared to Hatch Back car model.

Keywords
Flow Visualization, Sedan Car Model, Hatch Back Car Model, Flow Separation, Vortex.

I. Introduction
Aerodynamicist have a difficult task to perform, as they deal with invisible medium i.e. air, for their aerodynamic analysis. Greater understanding and valuable insights of the physical behavior such as boundary layer formation, flow separation and vortex generation etc., of an air flow can be known if the entire flow field was made visible by some technique which could be seen by our naked eyes or by a recording device. One such method is the flow visualization technique.

Flow visualization in air can be classified into surface flow visualization and off - the - surface visualization. Use of tufts, fluorescent dye, oil or special clay mixtures which are applied to the surface of a model comes under the category of surface flow visualization. These tufts, dyes and special coatings on the models are visually inspected as a function of time or after pre defined time duration. It is important to note that visualization that has been observed on the surface of the model cannot be an indicative for what is happening off the surface. In case of off the surface visualization foreign particles such as smoke particles, oil droplets or helium-filled soap bubbles are injected into the flow to follow the flow pattern. In the present study aerodynamic analysis by off – the - surface flow visualization technique is carried out for Sedan and Hatch back car models to checkout their aerodynamic efficiency.

II. Experimental Set Up
Experimental Analysis was conducted on two 1:30 scaled down rubber wood car models, out of which one is geometrically similar to commercially available Sedan car and the other model is a Hatch Back which is newly designed by modifying the 2d profile of the same Sedan car model. Both the car models have been fabricated such that they have same frontal area but varies only at the rear portion i.e., one is fabricated as a Sedan car model and other as the Hatch Back car model. Blockage ratio, found out with respect to projected frontal area of the car model in test section was about 3%, which is well below the permissible limit of 7.5% [1]. Flow visualization analysis was conducted for both the car models in a low speed open circuit suction type wind tunnel (Fig. 1) having the test section dimension of 390 × 302 × 1140mm.

(a)

(b)

Fig. 1: (a), (b). Suction Type Wind Tunnel

In the present study paraffin mineral oil was vaporized inside the smoke generator (Fig. 2) equipment to produce the smoke. Stream lines were generated in the wind tunnel by introducing the smoke produced from the smoke generator through small pipes placed in front of a test model. The choice of using smoke in a wind tunnel depends on several aspects such as; it should be dense, non toxic, non corrosive and should also have good viewing quality.

Analysis was carried out in the suction type wind tunnel at a free stream velocity of 1.5 m/s to achieve sharp streaks of smoke. Operating velocity was limited to 1.5 m/s due to the fact that at higher velocities the high speed air inside the wind tunnel would carry away the stream lines generated by the smoke generator there by forming mist like environment around the car model there by making the conditions unsuitable to take flow visualization photographs.

The laser beam passing through cylindrical lens produced the sheet of laser light. By using this light sheet, cross section of the wake formed around the car model was illuminated and the position of the vortices was located.

All the flow visualization photographs were recorded in Fujifilm Finepix S9600 DSLR Camera. The camera was placed at a distance of 0.45 m from the wind tunnel test section and at a height of 1.35 m from the ground.

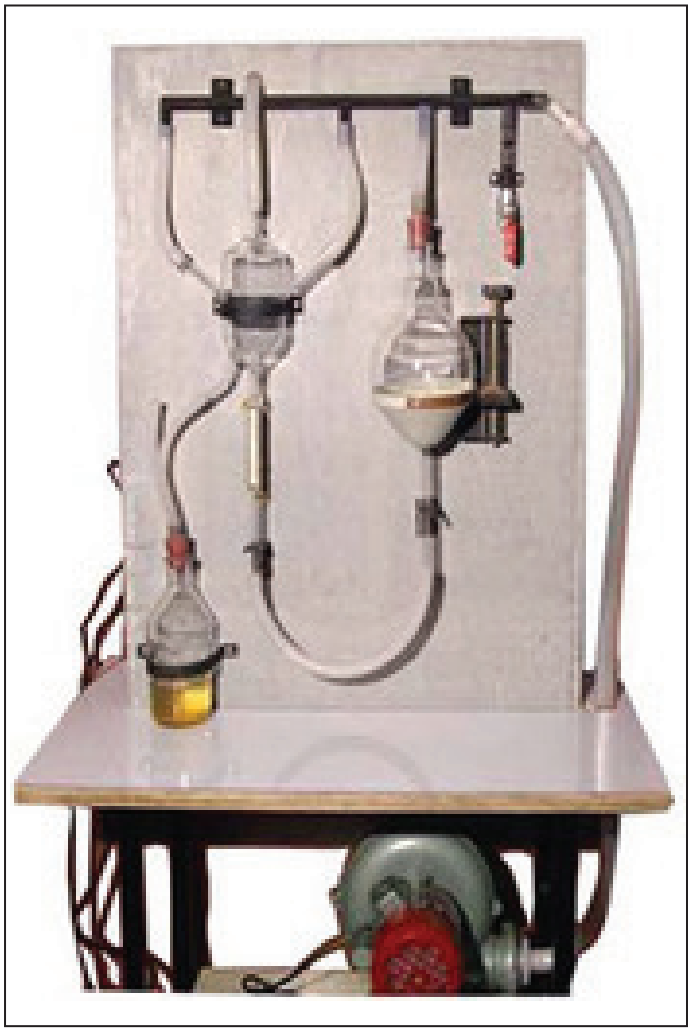


Fig. 2: Smoke Generator

III. Development of Flow Visualization Images

With proper experimental setup and operating conditions, laser illuminated steam line over the car models were recorded in a DSLR Camera. Recording device was set to capture the videos at 30 frames per second in order to capture even the slight variation in vortices formed at the rear end of the car model at different intervals of time. The recorded videos were converted into images or frames (Fig. 3) by using commercially available software. In the later stage these images were edited to get good quality flow visualization images. Vortices formed around the car models were highlighted by converting those images into grayscale (Fig. 4).

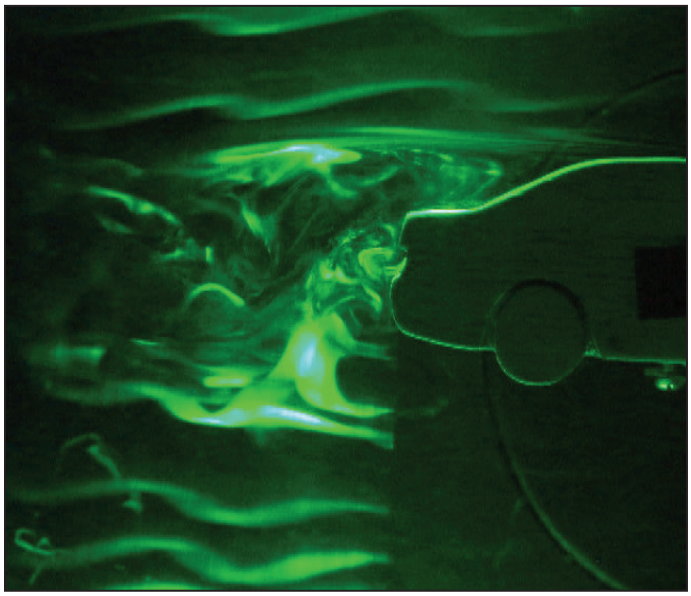


Fig. 3:

Fig. 4: Grayscale Image

IV. Resullts and Discussion

Fig. 5 shows the stream lined flow over front bumper region of Sedan car model and fig. 6 shows the stream lined smoke flow over front bumper region of Hatch Back car model. Since the front profile of both the car remains same, the flow over them also remains the same up to rear wind shield of both the cars. From the above flow visualization images it can be seen that the flow separates at the front bumper region and slips smoothly over the engine bonnet. As the flow moves further there is a slight vortex creation at the intersection of bonnet and front wind shield in both the car models, after which flow continues over the roof of both the car models and separates out at the roof and rear wind shield intersection.

ISSN : 2249-5762 (Online) | ISSN : 2249-5770 (Print)

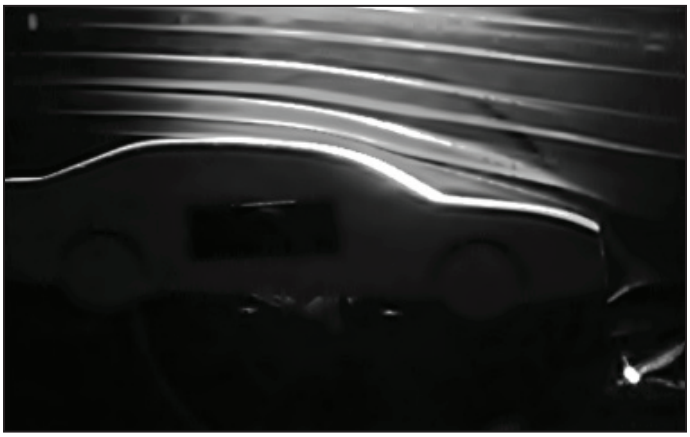


Fig. 5: Flow over Sedan Car Model

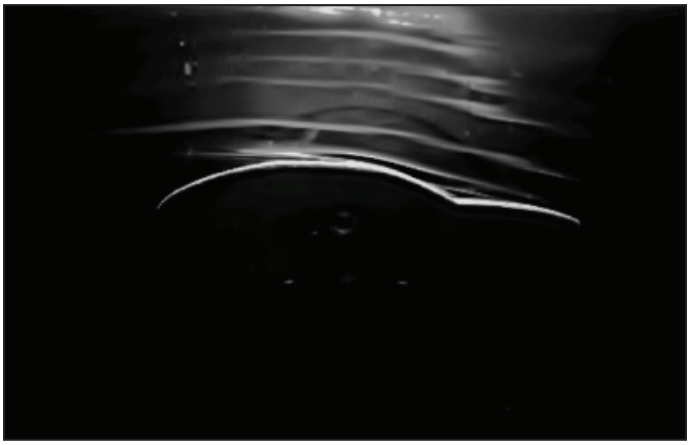
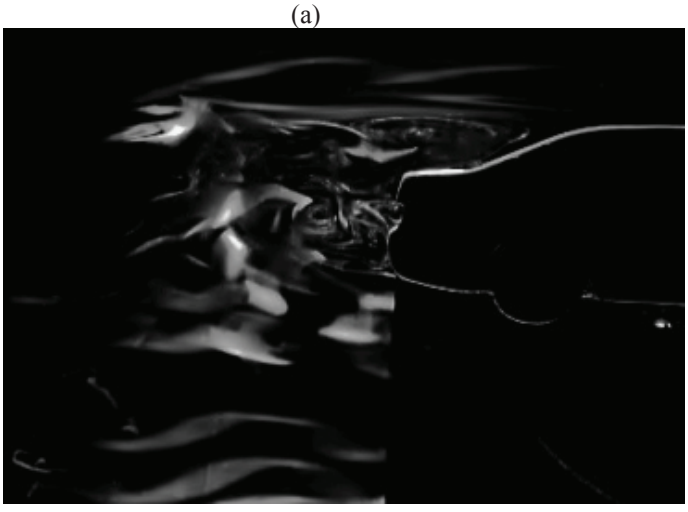
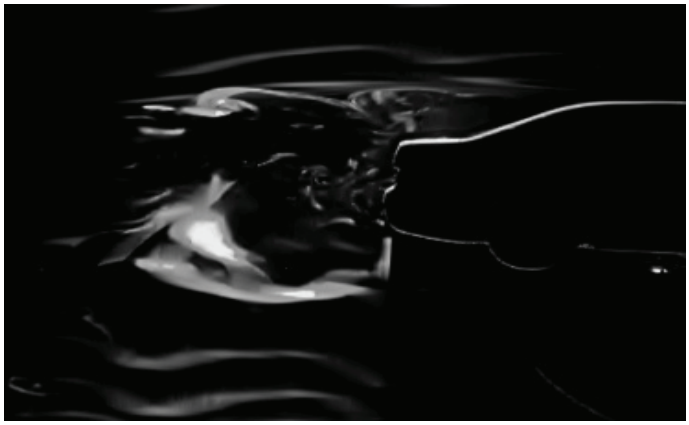


Fig. 6: Flow over Hatch Back Car Model

(a)



(b)



(c)

Fig. 7: (a) (b) (c) Flow Seperation and Vortices formation at the rear end of Sedan Car Model

(a)

(b)

(c)

Fig. 8: (a) (b) (c) Flow Seperation and Vortices formation at the rear end of Hatch Back Car Model

Fig. 7 and fig. 8 shows the flow separation and vortex generated at the rear end of Sedan and Hatch Back car models respectively at different instant of time at a velocity of 1.5 m/s. It can be seen that flow separates at the intersection of roof and wind shield in both the car models. This results in the formation of wakes or vortices at the rear end. Another reason for formation of wakes and vortices at the rear end is because of the vacuum created at the back of the car model as it moves forward. Due to the presence of boot in the Sedan car model, there is a slight flow reattachment in the boot region which reduces the negative pressure area left behind by the car model. Also it is clear from the fig. 7 & fig. 8 that wake formation is more in case of Hatch Back car model as compared to Sedan car model. This is due to the fact that hatch back car model has more negative pressure area and experiences more negative pressure at the rear as compared to sedan car model. So it can be concluded that Sedan car model is more stream lined, aerodynamic and experiences less drag force as compared to Hatch Back car model.

References

[1] William H. Bettes,"The Aerodynamic Drag of Road Vehicles: Past, Present, and Future", Engineering & Science, January 1982.

[2] Bhagirath Zala, Dr. Pravin P. Rathod, Sorathiya Arvind S, H. I. Joshi,"Comparative assessment of drag force of Hatch Back and Sedan car model by experimental method", International Journal of Advanced Engineering Research and Studies, Vol. 1, Issue 3, pp. 181-183, 2012.

[3] Bhavini Bijlani, Pravin P. Rathod, Arvind S. Sorthiya, "Experimental Investigation of Aerodynamic Forces on Sedan, Fastback and Square-Back Car by simulation in CFD" Review Study", International Journal of Emerging Technology and Advanced Engineering, Vol. 3, Issue 2, 2013.

[4] Sharath Kumar S N, Dr. C. K. Umesh,"Analysis of External Aerodynamics of Sedan and Hatch Back Car Models Having Same Frontal Area by Experimental Wind Tunnel Method", International Journal of Engineering Development and Research (IJEDR), Vol. 3, Issue 4, pp. 812-816, 2015.

[5] Sharath Kumar S N, Dr. C. K. Umesh,"Analysis of External Aerodynamics of Sedan and Hatch Back Car Models Having Same Frontal Area by Computational Method", International Journal of Research in Mechanical Engineering & Technology, IJRMET Vol. 6, Issue 1, Nov 2015-Apri l 2016.

[6] Yunus A Cengel, John M Cimbala,"Fluid Mechanics Fundamentals and Applications", McGraw-Hill Publications, 2006.

[7] Vehicle Aerodynamics: Hybrid Electric Vehicle Team, College of Engineering, San Diego State University, [Online] Available: http://kahuna.sdsu.edu/~hev/aerodyn.html (Accessed on 12-01-2015).

[8] Sharath Kumar S N,"Analysis of external aerodynamics of Sedan and Hatch back car models by Experimental and Computational Methods", M. E Thesis submitted to Department of Mechanical Engineering, University Visvesvaraya College of Engineering, Bangalore, India, 2015.

[9] Dinesh Dhande, Manoj Bauskar,"Analysis of Aerodynamic Aspects of SUV by Analytical and Experimental Method", International Journal of Emerging Technology and Advanced Engineering, Vol. 3, Issue 7, 2013.

[10] R. B. Sharma, Ram Bansal,"CFD Simulation for Flow over Passenger Car Using Tail Plates for Aerodynamic Drag Reduction", IOSR Journal of Mechanical and Civil Engineering, Vol. 7, Issue 5, pp. 28-35, 2013.

[11] Pramod Nari Krishnani,"CFD study of drag reduction of a generic sport utility vehicle", M. S Thesis submitted to Department of Mechanical Engineering California State University, Sacramento, 2009.

[12] Tank Nilesh R, R. Thundil Karuppa Raj,"Numerical Simulation of Aerodynamic forces acting on Passenger Vehicle While Overtaking", Research Journal of Recent Sciences" Vol. 1, 2012.

[13] Sasitharan Ambicapathy, J. Vignesh, P. Sivaraj, Godfrey Derek Sams, K. Sabarinath, V. R. Sanal Kumar,"3D Numerical Studies on External Aerodynamics of a Flying Car", World Academy of Science, Engineering and Technology, International Journal of Mechanical, Aerospace, Industrial and Mechatronics Engineering, Vol. 8, Issue 5, 2014.

[14] Manikandan. M, Shiva Prasad U, Ashish Ashok Suvarna, Anuj Bhat B, Vinayak Nair, Gunda Shivakrishna,"External Aerodynamic Analysis of TATA Nano using Numerical Tool", International Journal of Current Engineering and Technology, Issue 2, pp. 393-396, 2014.

[15] Deepak Kumar Kalyan, A.R. Paul,"Computational Study of Flow around a Simplified 2D Ahmed Body", International IJRMET Vol. 6, Issue 1, Nov 2015-April 2016. [Online] Available: http://www. i j r m e t. c o m. International Journal of Research in Mechanical Engineering & Technology, Journal of Engineering Science and Innovative Technology, Vol. 2, Issue 3, May 2013.

[16] Rehan Salahuddin Khan, Sudhakar Umale,"CFD Aerodynamic Analysis of Ahmed Body", International Journal of Engineering Trends and Technology, Vol. 18, No. 7, 2014.

[17] Manan Desai, S. A. Channiwala, H. J. Nagarsheth,"A comparative assessment of two experimental methods for aerodynamic performance evaluation of a car", Journal of Scientific and Industrial Research, Vol. 67, pp. 518-522, 2008.

Sharath Kumar S N received his B.E. degree in Mechanical Engineering from Visvesvaraya Technological University, Belgaum, India in the Year 2013. He completed his Post graduation in University Visvesvaraya College of Engineering, Bangalore University, Bangalore, India with specialization in Thermal Science and Engineering in the year 2015. Currently he is working as Assistant professor in Department of Mechanical Engineering, Dr. Ambedkar Institute of Technology, Bangalore, India. His research interests include Vehicle Aerodynamics, Turbine Blade cooling and Heat exchangers.

ISSN : 2249-5762 (Online) | ISSN : 2249-5770 (Print)

Dr. Sathish S is currently working as Assistant Professor in the Department of Mechanical Engineering, Dr. Ambedkar Institute of Technology, Bangalore, India, with more than 19 yrs teaching experience. He has published 17 International Journals. He is currently guiding 4 Doctorate students.