International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 08, August-2016

Wake Visualization Behind Different Vehicle Body Shapes

Gurjot Kaur
Amity School of Engineering and Technology
Noida, U.P, India

***Abstract* — With the increasing fuel prices and more strict requirements in CO_2 exhaust levels, it is necessary to reduce the energy consumption of vehicles. Since aerodynamics plays an important role in total driving resistance of vehicles, it is logical step to focus in this area. A large part of the aerodynamic air resistance of vehicles can be contributed to the pressure drag caused by its wake. The problem of Wake visualization in fluid mechanics is of considerable interest, especially the visualization of the complex flows, which is used to verify the existing theories. The flow region which presents the main contribution to a car's drag, and which poses severe problems to numerical predictions and experimental studies, is the wake flow behind the car. The location where the flow separates decides the extent of the separation zone, and consequently the drag force. When the air passes over the vehicle surface, it causes the air over the surface change its behavior, resulting in a low pressure region and a high pressure region. This pressure difference along with vortex shedding causes drag thereby increases the fuel consumption. Clearly, a more precise simulation of the wake flow and of separation process is essential for the correctness of drag predictions. Three different car models and a bus shape model are used for experiments and Experiments are performed at different angles.**

Keywords: Flow visualization, Aerodynamics, Wind Tunnel, Smoke method

1. INTRODUCTION

Flow visualization or flow visualization in fluid dynamics is used to make the flow patterns visible, in order to get qualitative or quantitative information on them. Flow visualization is the art of making flow patterns visible. Most fluids (water, air, etc.) are transparent, and their flow patterns are invisible to us without some special methods to make them visible. These experimental methods include methods, like say spilling ink into water.

The problem of flow visualization in fluid mechanics is of significant interest, especially the visualization of complex flows, which is used to verify the existing theories. The flow region which presents the major contribution to a car's drag, and which poses problems to the numerical predictions and experimental studies as well, is the wake flow behind the car. The location where the flow separates determines the size of the separation zone, and consequently the drag force. Clearly, a more precise simulation of the wake flow and of the separation process is essential for the correctness of drag predictions. However, a real-life automobile is a very complicated shape to model or to study experimentally. In order to realize the physical phenomena of the wind flow over typical launch vehicle, flow was simulated using Smoke tunnel.

2. METHODOLOGY ADOPTED

1. Literature review.
2. Procurement of specimens.
3. Experiments performed on Smoke Tunnel using different models of bus & cars.
   - Flow visualization of Hatch Back shape Model
   - Flow visualization of Square Back shape Model
   - Flow visualization of Sedan Model
   - Flow visualization of Bus shape Model
   - Flow visualization of these above given models at different angles
4. Analysis and comparison based on testing on different models.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 08, August-2016

3. SMOKE TUNNEL USED FOR EXPERIMENTS

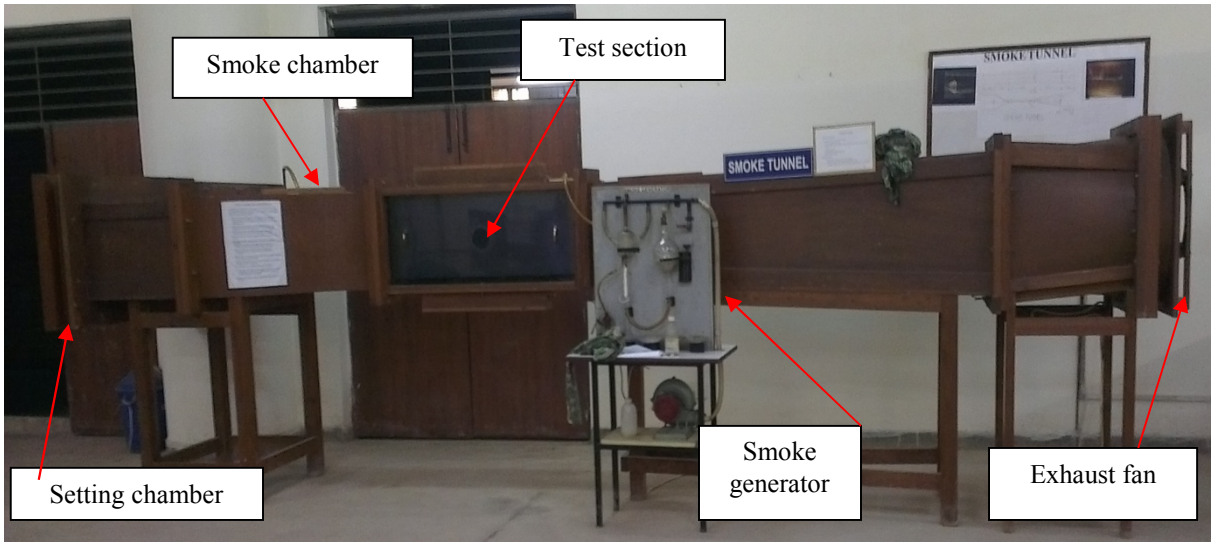

Figure 1 Smoke Tunnel used for experiments

Figure 2 Test Section of Smoke Tunnel

Figure 3(a) Exhaust Fan at suction end, Figure 3(b) Setting Chamber

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 08, August-2016

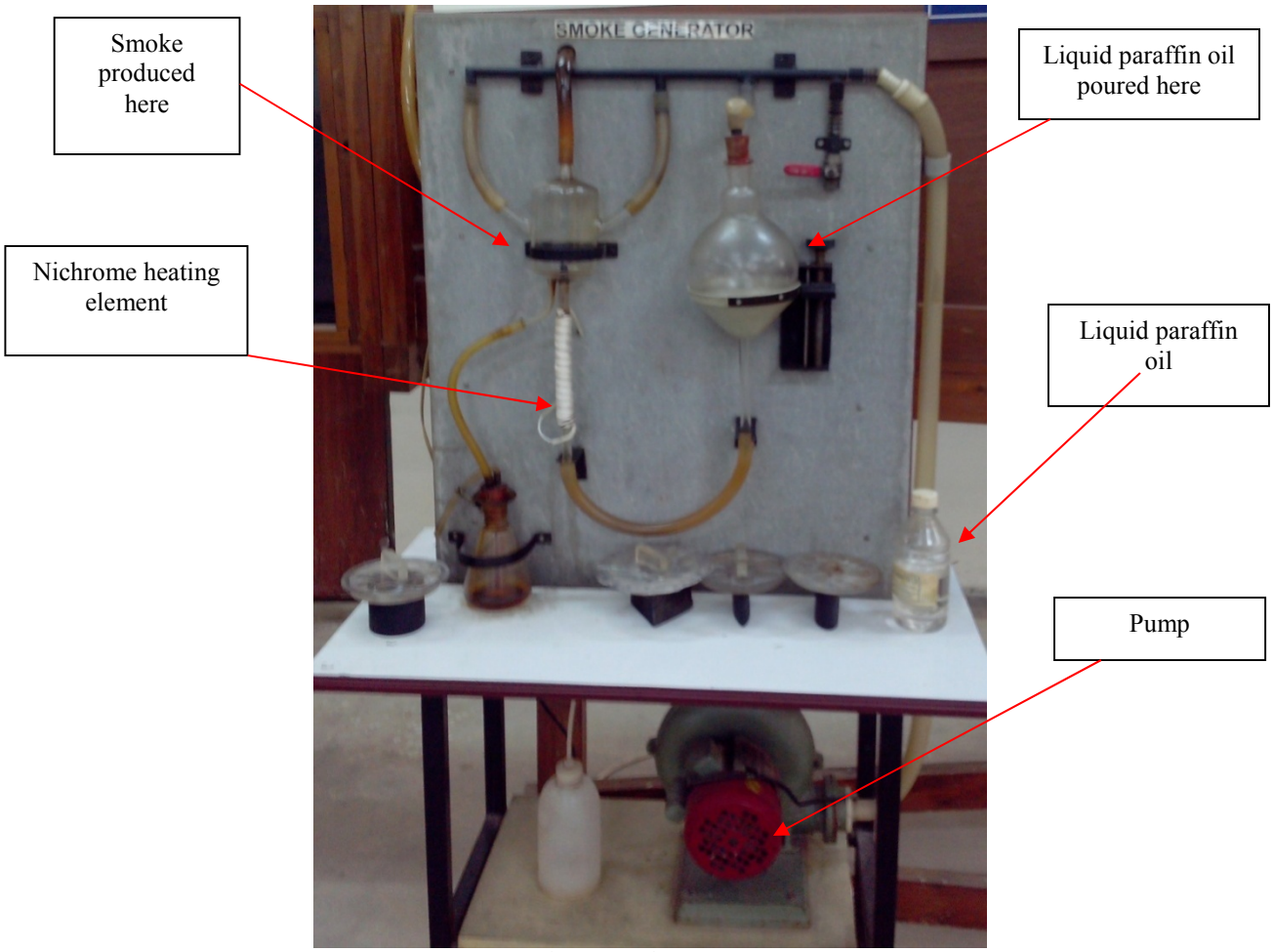

Figure 4 Smoke Generator

4. DIMENSIONS OF SMOKE TUNNEL

- Test section of 45cm*5cm*120cm size.
- Exhaust fan at suction end
- Setting chamber of 45cm*80cm*60cm with honeycomb section and wire mesh
- Contraction -16.
- Overall length of Wind Tunnel=6m.
- Suitable smoke generator.

5. EXPERIMENTS

5.1. Objective

Flow Visualization of models of car (Hatchback), wooden model of sedan car, model of square back car, and model of bus in a Smoke Tunnel at different angles

5.2. Material Used

Acrylic sheet, Round Protector, Liquid Paraffin Oil, Fevi Bond, Specimen of car models and bus model, Digital Camera

5.3. Equipment Used

- Smoke Tunnel.

5.4. Procedure

Following models taken for experiments:-

- Size of hatchback model taken : Width - 5cm, Length-10cm
- Size of square back model taken: Width - 4cm, Length- 8cm.
- Size of wooden sedan car model taken : Width - 4.5cm, Length-9.3cm
- Size of bus model taken: Width - 3cm, Length - 9cm.

5.5. Experiments performed on Smoke Tunnel.

1. Models are glued on acrylic sheet and acrylic sheet is glued on round protector.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 08, August-2016

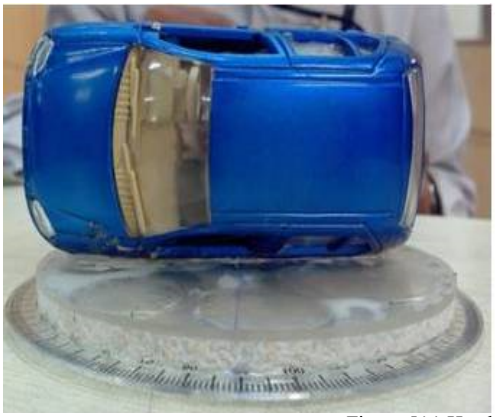

Figure 5(a) Hatchback model glued on acrylic sheet
Figure 5(b) Squareback model is glued on acrylic sheet

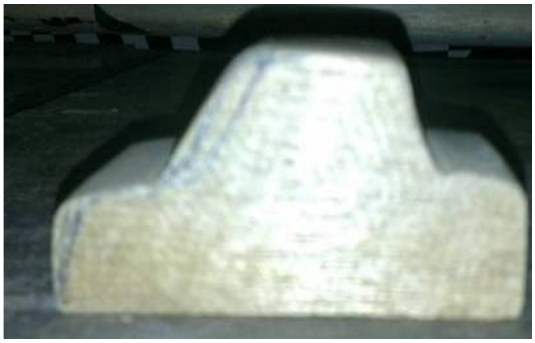

Figure (6) Wooden model of sedan

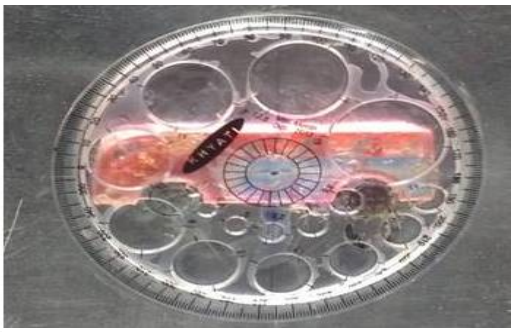

Figure 7 Model of bus

2. *Models fitted in test section of smoke tunnel.*

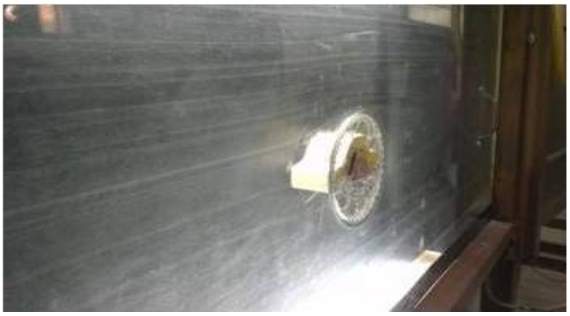

Figure 8(a) Hatchback model fitted in test section of Smoke Tunnel
Figure 8(b) Wooden model of sedan fitted in test section

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 08, August-2016

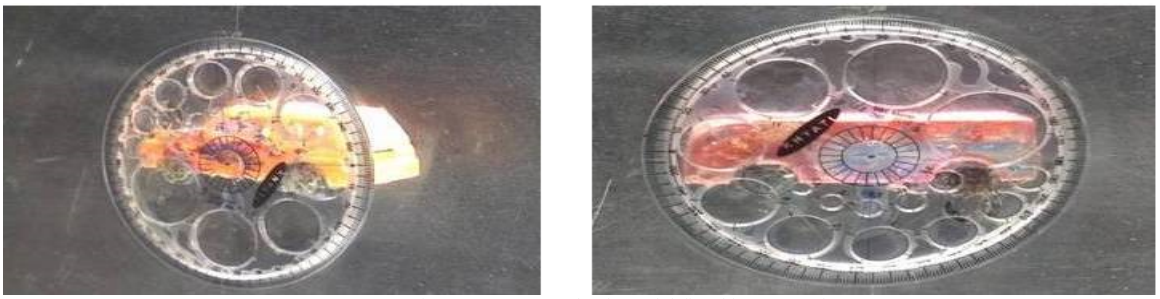

Figure 9(a) Square-back model fitted in test section
Figure 9(b) Bus model is glued on acrylic sheet

3. Liquid paraffin oil is poured to container in smoke generator.
4. Liquid comes in contact with Heating element (Nichrome) covered on glass tube.
5. When liquid reaches at evaporating temperature than smoke generates.

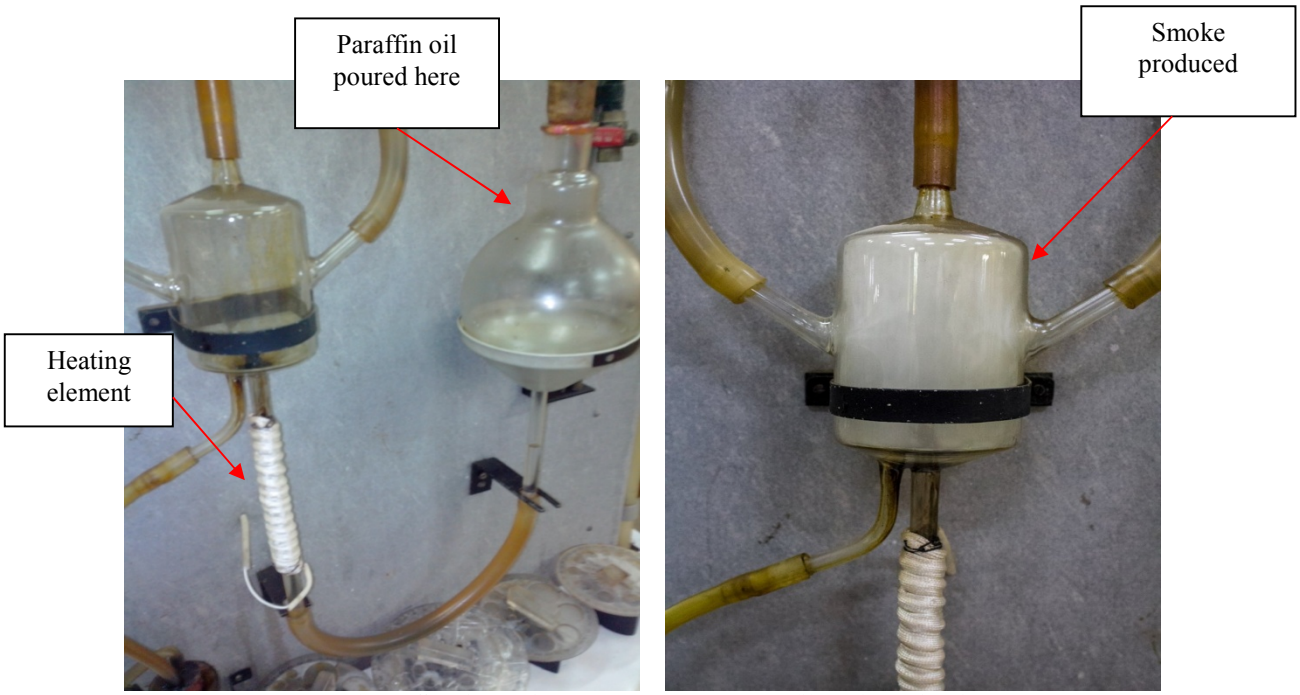

Figure 10(a) Paraffin oil poured in Container.

Figure 10(b) Smoke produced

6. The smoke is forced by pump to the smoke chamber.
7. By suction the smoke is sucked by adjust fan at end.
8. Flow visualization is done at different angles.
9. Pictures taken while doing experiment.

6. OBSERVATIONS

6.1. Observations when experiment performed on hatchback car.

Figure 11(a) Flow visualization of Hatchback Shape model in Smoke Tunnel

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 08, August-2016

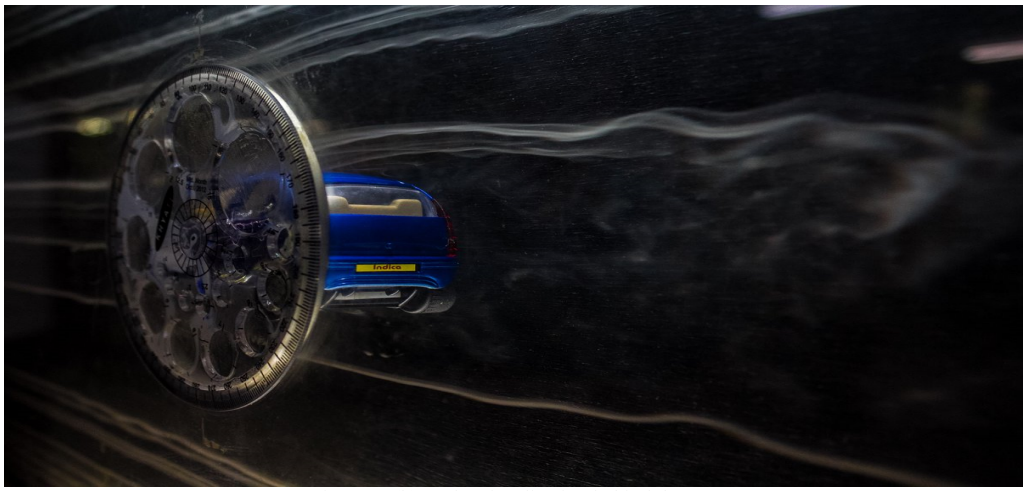

Figure 11(b) Wake visualization behind the car

Figure 11(c) Wake visualization at positive 15 degree angle.

Figure 11(d) Wake visualization behind the car at positive 15 degree angle.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 08, August-2016

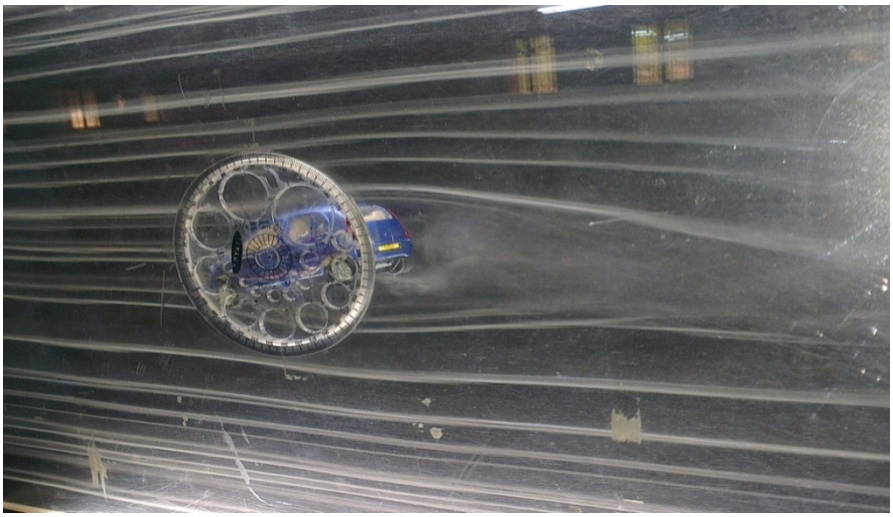

Figure 11(e) Wake visualization at negative 10 degree angle.

6.2. Observations when experiment performed on wooden sedan car.

Figure 12(a) Flow visualization of sedan model

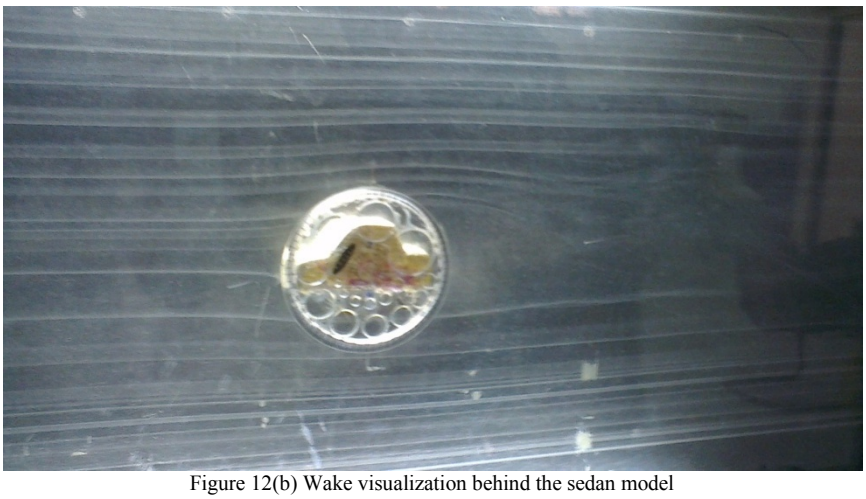

Figure 12(b) Wake visualization behind the sedan model

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 08, August-2016

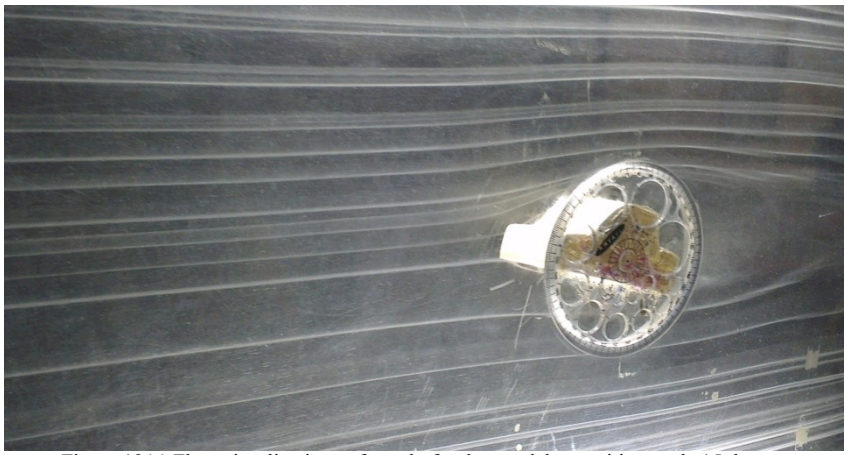

Figure 12(c) Flow visualization at frontal of sedan model at positive angle 15 degree

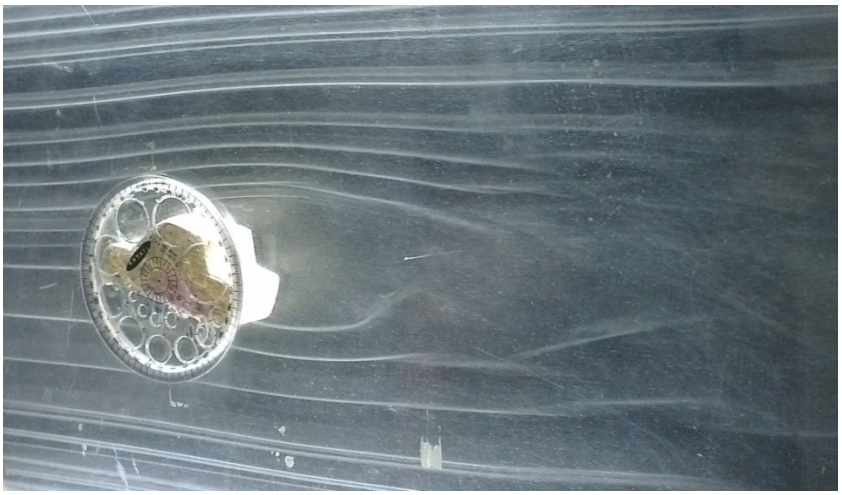

Figure 12(d) Flow visualization behind the sedan model at positive angle 15 degree

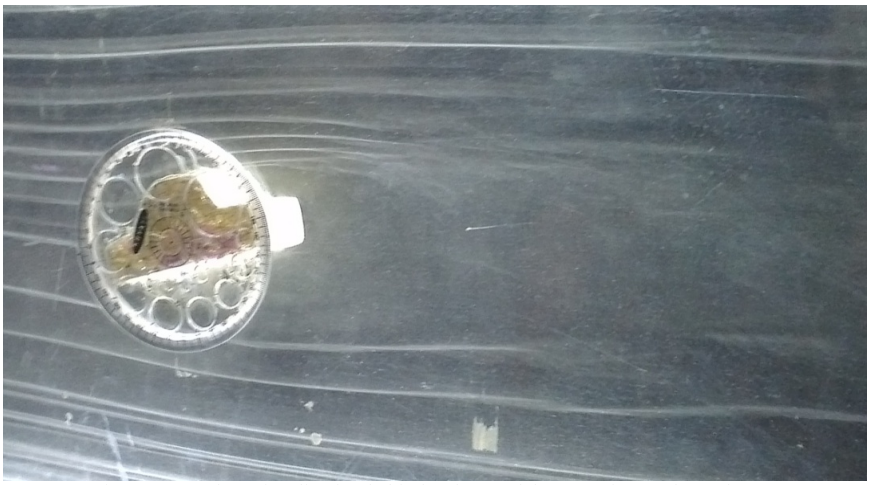

Figure 12(e) Flow visualization behind sedan model at negative angle 10 degree

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 08, August-2016

Figure 12(f) Flow visualization of sedan model at negative angle of 10 degree

6.3. Observations when experiment performed on square-back car.

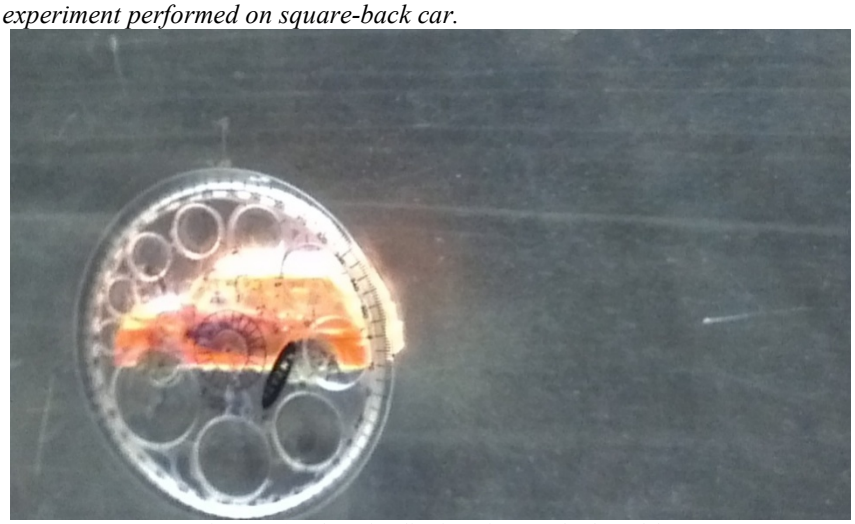

Figure 13(a) Flow visualization of Square-back model

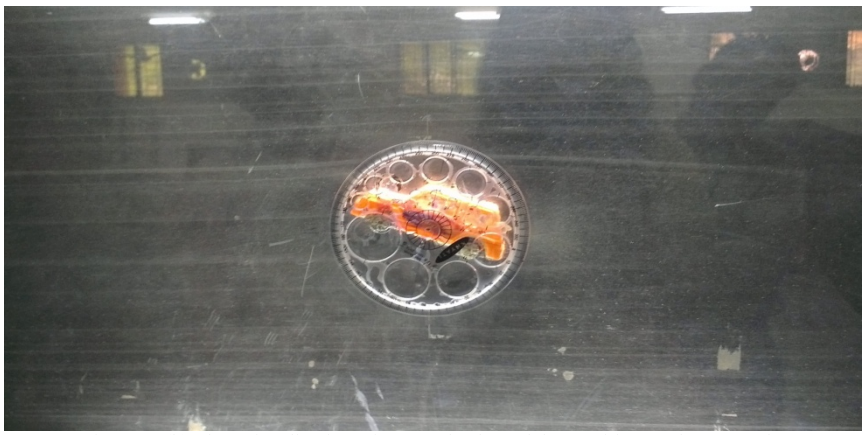

Figure 13(b) Flow visualization of Square-back model at positive 15 degree angle

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 08, August-2016

Figure 13(c) Flow visualization behind the Square-back model at negative15 degree angle

6.4. Observations when experiment performed on model of bus.

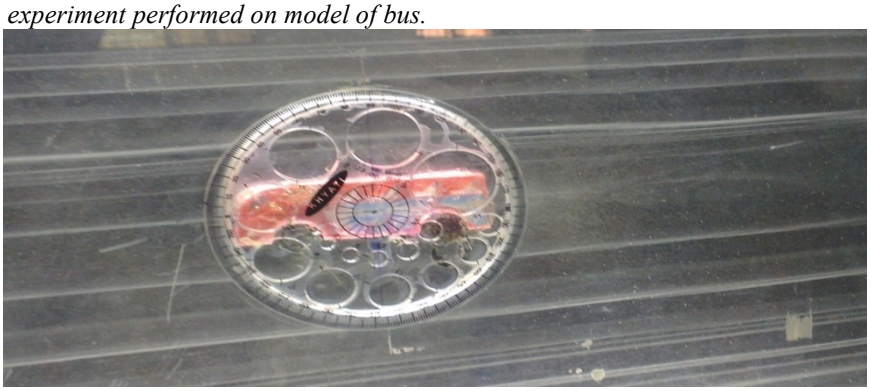

Figure 14(a) Flow visualization of Bus model in test section

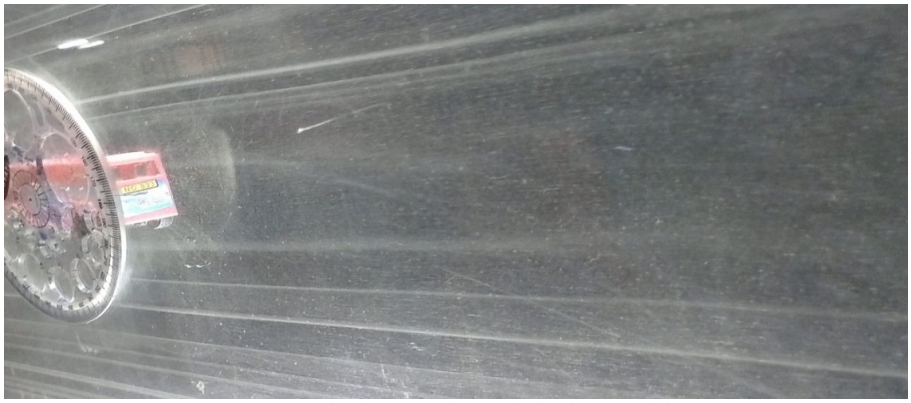

Figure 14(b) Flow visualization behind Bus model

Figure 14(c) Flow visualization of Bus model at positive 15 degree angle

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 5 Issue 08, August-2016

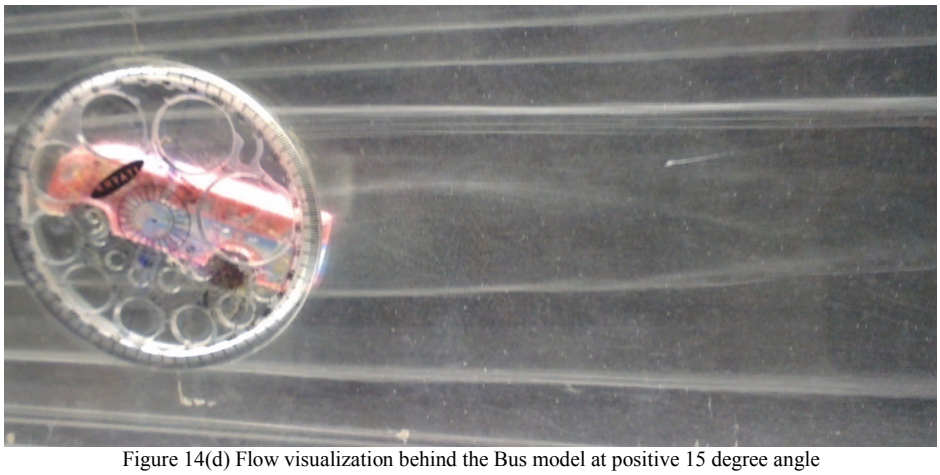

Figure 14(d) Flow visualization behind the Bus model at positive 15 degree angle

Figure 14(e) Flow visualization behind the Bus model at negative 10 degree angle

7. RESULTS

Wake structure and wake region changes with different shapes of models and wake structure also changes for the same model at different angles. Wake visualization of different models is done at positive and negative angles. Wake visualization is very important for drag predictions.

8. CONCLUSION

The flow region which represents the major contribution to a car's drag, and which posses severe problems to numerical predictions and experimental studies as well, is the wake flow behind the car. Therefore, a better understanding of the flow structures in the wake is a necessary in order to develop methods for drag reduction. Investigations on different vehicle models are conducted using Smoke Tunnel testing.

A more exact simulation of the wake flow and of the separation process is essential for the correctness of drag predictions. However, a real-life automobile is a very complex shape to study experimentally. Three different car models and a bus shape model are used for experiments and Experiments are performed at different angles. In order to understand the physical phenomena of wind flow behind the different models the flow is simulated using smoke tunnel.The critical shapes can have significant unsteady effects. The different shapes can cause sudden changes in size and structure of wake and corresponding large changes in drag. Wake visualization behind the different models of car and bus shape model is done in this project.

REFERENCES

1) Katz, J., Race Car Aerodynamics. Robert Bently Publishers, Cambridge, MA, 1995.
2) Gregor Seljak, Race Car Aerodynamics, April 8, 2008.
3) W-H Hucho: Aerodynamics of Road Vehicles.
4) H. Lienhart, C. Stoots and S. Becker,"Flow and Turbulence Structures in the Wake of a Simplified Car Model (Ahmed Model)", Lehrstuhl für Strömungsmechanik (LSTM)Universität Erlangen-Nürnberg Cauerstr. 4, 91058 Erlangen, Germany.
5) Selvi Rajan.S, Santhosh kumar. M and Lakshmanan. N, "CFD Analysis and Wind Tunnel Experiment on a Typical Launch Vehicle Model", Tamkang Journal of Science and Engineering, Vol. 12, No. 3, pp. 223_229 (2009)
6) K.C. Chung, "Wind Tunnel Experiment on the effect of Wind on Smoke Exhaust Systems for a high rise building", Mechanical Engineering, National Yunlin University of Science and Technology Touliu City, Yunlin 640, Taiwan
7) Shahrin hisham Amirnordin, Wan saiful-islam wan salim, Suzairin Md Seri and Mohd Faiz Ariffin, "The effects of Air flow in The wake of a large Vehicle on Trailing a Passenger car", Journal of Engineering and Technology, ISSN: 2180-3811, Vol. 2, June 2011
8) Marchant, E.W., "Effect of Wind on Smoke Movement and Smoke Control Systems", Fire Safety Journal, 7, 55-63, 1984.
9) Mathieu Roume´as, Patrick Gillie´ron, and Azeddine Kourta, "Analysis and control of the near-wake flow over a square-back geometry".
10) L. Sterken, S. Sebben and L. Löfdahl, "Alternative Approach in Ground Vehicle Wake Analysis", World Academy of Science, Engineering and Technology 68 2012.