Flow visualization over an Indian auto-rickshaw model

Saransh Abbey, Pratyush Kumar Singh, Neha Singh and Kannan B T
Department of Aerospace Engineering, SRM Institute of Science and Technology, Kattankulathur, Kancheepuram District, Tamil Nadu, 603203, India.

E-mail: saranshabbay_harish@srmuniv.edu.in, pratyushkumarsingh_pramod@srmuniv.edu.in, nehasingh_jaishankar@srmuniv.edu.in, skyinventorbt@gmail.com

Abstract. From the late twentieth century, spark-ignition engine powered Indian auto-rickshaw is treated as the most common mode of transportation for rural and urban area activities. This three-wheeler vehicle is found to be an economic mode of transport for shorter distances. This study focuses on the flow visualization over the vehicle using smoke based visualization technique. The visualization is achieved by using a monochromatic laser sheet on the vehicle with seeding particles at various operating conditions. This visualization study helps researchers to understand the external flow physics. The results in terms of flow pattern obtained with the visualization gives an understanding of the wake formation by the vehicle. The results also indicate the presence of complex flow around the vehicle.

1. Introduction
Being the fourth largest in the world of sales, India has grown the automobile industries with an increment of 9.5 percent year by year and also took a step in export business. Most of the Indian markets have demanded the need for two-wheeled vehicles to three-wheeled vehicles for a long period of time. One of the famous Indian automobile vehicle–Auto rickshaw is the main focus of this paper. There has been numerous study of flow physics of different road vehicles like SUV, Sedan Cars and even Motorbikes and Cabriolets that run at various speed which gave a huge contribution to various automobile companies about the flow behaviour at various section of the body.

2. Literature Survey
Considering the vehicle body in close proximity with the ground, W. Hucho [1]. gave detailed understanding regarding the aerodynamics aspects of the on-road vehicles with proper experimental pictures of the experiments conducted by him and the contributors of the book. They implemented the general aerodynamic theory over the bluff body to various road vehicles developed over the period of time. T. Han [2]. did his computational study of a three-dimensional turbulent flow around the bluff body. The Reynolds Navier-Stokes Equation with k-epsilon equation for turbulence flow with Reynolds number of $4.3 \times 10^6$. The numerical simulation conducted with a second-order discretization scheme for reducing the numerical diffusion. The flow resulted in a reverse flow and trailing edge vortex formation around the body. Williams D B and Dominy R G [3]. conducted two different experiments i.e. by introducing the twin hot wire probes at the wake region and frequency domain correction method to understand the unsteadiness of the passenger car with the effect on stability. The unsteadiness was resulted because of the separation bubble at the close proximity of the model and also due to the large radius curved rear surfaces. Abdel Azim and Abdel Gawad [4]. conducted a flow visualization study of the interferences caused when two passenger cars are very close to each other.
The resulted flow was further justified with the pressure distribution and the drag forces. The main factor for the resulted values or the flow properties during the experiment was the Reynold number according to the authors. Bayraktar I et.al [5]. did a numerical and experimental study on the ground vehicle Aerodynamics of the Ahmed body. The experimental and the numerical setup were compared on the basis of various back angles with the variation of Reynolds number.

The comparison of both the data resulted in difference of the lifting force created during the simulation of the body. Howard R et al [6]. did a large eddy simulation over an Ahmed body which gave a complex result of the flow over the body with respect to time. The priceless/trio_o platform was used with a slant angle of 28 degrees for the simulation. Knight J et.al [7]. did his study of the rooftops of the convertible cars with the help of numerical and experimental methods. The results of the study show the key elements of the methodology that could be used in designing the convertible cars. Cernat M and Cernat A [8]. did his study regarding the lifting surface over the body of the vehicle. The study gave a detailed understanding of the flow around the body due to the interaction of the various components resulting in the vortex shedding[9]. Guo Z et.al where optimization of the hatchback cars resulted in a reduction of the drag and rear drag by 5.64 and 7.21 % respectively.

Researchers know the aerodynamic forces play an important role in the vehicle which is the interaction of various vortex formation and flow separation. The main motive of this study is to understand the flow around the scaled-down version of the actual Bajaj auto rickshaw experimentally. The flow of the free stream is at a various low-speed Reynolds number in a smoke wind tunnel. This study is carried out at Visualization Laboratory, Aerospace Hangar, SRM Institute of Science and Technology, Chennai, India. This study helps to understand the vehicle structure in detail and make some modifications that are necessary to it.

3. Experimental Methodology
The experiments were successfully conducted using a subsonic smoke tunnel as shown in figure 1. The test-section of the tunnel had a rectangular cross-section area and a length of 0.184 x 0.10 m² and 0.312 m respectively. The flow speed of the wind tunnel could be adjusted using the motor which varies the rpm of the turbine to let it run at various speeds. With the Reynolds number varying for the scale down version from Re=6000 to Re = 30000, where the Reynolds number is dependent on the length of the auto-rickshaw.

Figure 1. The subsonic smoke tunnel.
The scaled-down model used for the flow visualization is in the ratio of 1:14 to the actual Bajaj Auto as shown in figure 2 which are most likely to be seen on the Indian roads. The proper dimension of the auto-rickshaw in terms of length, height and width are 10 x 8 x 5 cm respectively. To understand the flow characteristics and for the visualization, the test-section wall was made out of clear glass to see the smoke flow. The test bed was injected with the smoke generated by the paraffin smoke generator as shown in figure 3. The continuous flow of the smoke gave a flow pattern around the vehicle body. The flow was illuminated by the monochromatic laser (532nm) from the top face of the test section. The behaviour of the flow was captured by the digital camera having up to 920 fps with 12 megapixels.

4. Flow study at various speeds
The experiment conducted on the auto-rickshaw kept in the direction of the upcoming flow was visualized at different velocity i.e. 1 m/sec, 2m/sec, 3 m/sec, 4 m/sec and 4.5 m/sec corresponding to the range of the Reynolds number between 6000 to 30000. The flow separation on the body was examined carefully. To carefully understand the flow conditions, the auto model faces are represented as windward face, upper surface, and leeward face.
4.1 Flow observation at 1m/sec

Form the Figure 7 shown, the flow structure around the scaled-down auto-rickshaw is very easily comprehensible. The streamline flow of the smoke in the windward face of the model seems to get separated at an approximate height of ¼ of the actual model height from the top and the rest of the flow turns down and flows through the bottom of the auto-rickshaw. Also at the upper surface of the auto, the flow remains attached to them throughout the surface length and gets separated at the leeward face of the auto body forming a low-intensity vortex region and easily gets dissipated in the downstream flow. Over the lower rear region of the auto-rickshaw, a low-pressure region is created where the flow from the lower side of the auto-rickshaw flows through and fills up with the smoke particles. The most down force created by the high-pressure region formed at the back of the auto and the drag force can be determined by the low-pressure region created by it.

4.2 Flow observation at 2m/sec

As the upstream flow speed increased, the properties of the flow also changes over the body. The figure 8 shows the flow around the auto model at 2 m/sec. On comparison with the flow at 1m/s, the flow at the windward face of the auto remains the same but over the upper surface, the flow does not get attached to the surface of the auto body for 80% of its length which created a new low-pressure region over the upper body. With relation to the velocity and pressure, the smoke particles in this region are high than the upstream until the flow gets attached to the body and creates instability in the flow. This instability inflow resulted in the vortex formation at the leeward face of the body. The intensity of the vortex shedding was high enough to easily get captured by the digital camera when illuminated with the laser sheet. Over the bottom of the auto, the flow properties did not change visually and resulted in creating a low-pressure region at the bottom of the leeward face of the body. The high pressurized flow from the upper leeward face flows to the lower part of which tends to change the flow direction downwards as represented in Figure 8.
4.3 Flow observation at 3m/sec
As the speed of the flow increases, the separation region over the body is seen to shift in the front area of the upper surface as seen in Figure 9. This forms a separation bubble at the top and the flow gets reattached to the surface. This separation region is very sensitive to the incoming laminar flow and changes the fluid property to turbulent nature. The turbulent flow where it reattached to the body is referred to as a reattachment point, which can be estimated by about 10% of the length from the front of the body. The volume covered by the region of the separation of laminar to turbulent flows also called a laminar separation bubble. The flow inside the separation bubble is circular in nature continuously. The turbulent flow got reattach to the body and flew at a higher speed which lead to a huge amount of the skin friction drag over the body. The formation of vortex shedding behind the auto does not take place due to high-speed flow, also the low-pressure region at leeward face decreases as compared to the 1m/sec and 2m/sec. Also, it is seen that the first laminar streamline mergers with the upper layer due to the turbulence created by the primary smoke layer.

4.4 Flow observation at 4m/sec
Figure 10 shows most of the same traits of the flow structure as seen for the case of the flow velocity of 3m/sec. The separation bubble length slightly gets increased, which can be clearly seen by comparing the flow pattern from the previous case. The transition from laminar to turbulent flow occurs almost about the mid-length of the body. This turbulence inflow leads to the merging of the adjacent streamline flowing over the upper body creating a higher skin friction drag coefficient after the formation of the bubble. The rest of the flow from high to low-pressure region at the leeward face.

4.5 Flow observation at 4.5m/sec
The Figure 11 represents the flow structure of the scaled model kept at the free stream velocity of 4.5 m/sec. The figure clearly shows that the separation bubble formation which was seen in the previous two cases but the length of the bubble is almost 50% of the length of the auto model. The primary layer of the streamline flow gets attached to the adjacent flow before the separation bubble formation. The reattachment point on the upper surface shifts almost 10-20% behind in comparison to the 3 m/sec. also at the leeward face of the auto, it is seen that the turbulent flow creates the smallest low pressure than all the previous visualization cases. The experiments conducted gave a brief idea of the flow around the actual auto-rickshaw and can be justified with the help of a detail numerical simulation of the same auto-scaled model. Also, the use of higher speed wind tunnels up to 10 to 15 m/sec can give a better result of the flow physics.
Figure 8. Side view of the flow structure over the auto-rickshaw at 2 m/s with separation zone at the upper surface. Illuminated by 1mm thick laser sheet.

Figure 9. Side view of the flow structure over the auto-rickshaw at 3 m/s with separation bubble formation over the upper surface at 10% of approximate length.
5. Conclusion
The visualization gave a brief idea about the flow structure of the flow around the different faces of the auto model at various speeds. Considering the flows, the vortex generation due to the 1:14 scale model was generated at lower speeds which were reduced at higher velocities and also the formation of the separation bubble at higher velocity on the upper surface of the body eliminated the vortex shedding at the leeward face which lead to formation of smaller low-pressure region. The experiments done to visualize the flow around the bluff body like auto-rickshaw which are the famous vehicle used on Indian roads gave a brief idea of the flow physics around it. Yet there can be some modification over the body of the model to reduce the drag values.
Acknowledgement
The corresponding author wishes to acknowledge the usage of the Flow Visualization laboratory for the experiments, Advanced Computing lab for the documentation and also the help rendered by the hangar supervisor and lab assistant for the video recording reported in this paper.

References
[1] Hucho W H 1981 Aerodynamics of Road Vehicles: From Fluid Mechanics to Vehicle Engineering English Edition
[2] Han T 1988 A Navier-Stokes analysis of Three-Dimensional turbulent flows around a bluff body in ground proximity 1st Natl Fluid Dyn. Conf. 27 1213–19
[3] Sims Williams D B and Dominy R G 1998 Experimental investigation into unsteadiness and instability in passenger car aerodynamics SAE Tech. Pap.
[4] Abdel Azim A F and Abdel Gawad A F 2000 A flow visualization study of the aerodynamic interference between passenger cars SAE Tech. Pap.
[5] Bayraktar I, Landman D and Baysal O 2001 Experimental and computational investigation of Ahmed body for ground vehicle aerodynamics SAE Tech. Pap.
[6] Howard R J A and Pourquie M 2002 Large eddy simulation of an Ahmed reference model J. Turbul. 3 37–41
[7] Knight J J, Lucey A D and Shaw C T 2010 A predictive capability for the aerodynamic deformation of convertible car roofs Proc Inst. Mech. Eng. Part D J. Automob. Eng. 224 1013–20
[8] Cernat M V and Cernat 2017 A Numerical study of aerodynamic effects on road vehicles lifting surfaces AIP Conf. Proc. 1798
[9] Guo Z, Zhang Y and Ding W 2019 Optimization of the aerodynamic drag reduction of a passenger hatchback car Proc Inst. Mech. Eng. Part G J. Aerosp. Eng. 233 2819–2836