Observation of Flow Dynamics Across A VW Beetle

Stephen Wong
MCEN 5151: Flow Visualization
Team Photo #1 Assignment
March 20, 2014

Table of Contents

Table of Figures

Introduction

The analysis of the flow dynamics associated with cars has been a constant challenge. The optimization of this particular aspect of automotive construction allows lower fuel costs, better performance, and the ability to allocate resources towards other areas needed for improvement [6]. In the following experiment, the use of a specialized Flow Tank, henceforth referred to as the FLUME, was used in order to capture the specific aspects associated with the fluid flow over a VW die cast model car. To greater emphasize the fluid dynamics associated with the VW beetle, blue dye was injected into the FLUME such that patterns below the surface could be more easily visualized. Additionally, the use of a relatively bright incandescent lighting fixture, specialized acrylic white background, and a custom built syringe for dye injection permitted the capture and creation of more complicated and turbulent flow patterns. The following report will detail the experimental setup, technique, and flow behavior associated with the VW model car, as seen below in Figure 1:

Figure 1-Application of Blue Dye in a FLUME for a VW Die Cast Model

During the execution of this experiment, several team members in addition to myself were involved in the equipment setup, equipment procurement, dye application, cleanup, reset operation, and camera operation. In this sense, I would like to acknowledge Professor Hertzberg, Zach Stein, Andrei Molchanov, and Nick Stites for helping me set up the photo, operate the FLUME, and take this wonderful image.

Experimental Setup

The following experiment utilized a specialized flow tank called a FLUME to capture the image. Originally, several different car models were purchased for use in the FLUME. Unfortunately, the FLUME channel width was 3.089,” which did not accommodate a majority of the model cars we wanted to use. Consequently, the only model that was capable of fitting in the FLUME itself was the VW beetle with dimensionality of 5.5” length by 3” width. Additionally, in order to keep the car from moving during flow, fishing line was tied around the axel of the front wheel and attached to the FLUME’s base via a metal hook that could be threaded through an opening at the bottom. A general overview of the experimental apparatus is shown below in Figure 2:

Figure 2-Basic overview of the FLUME used for the purposes of determining fluid dynamics

For the purposes of simplicity, a schematic of the car within the flume chamber is provided below, as seen in Figure 3:

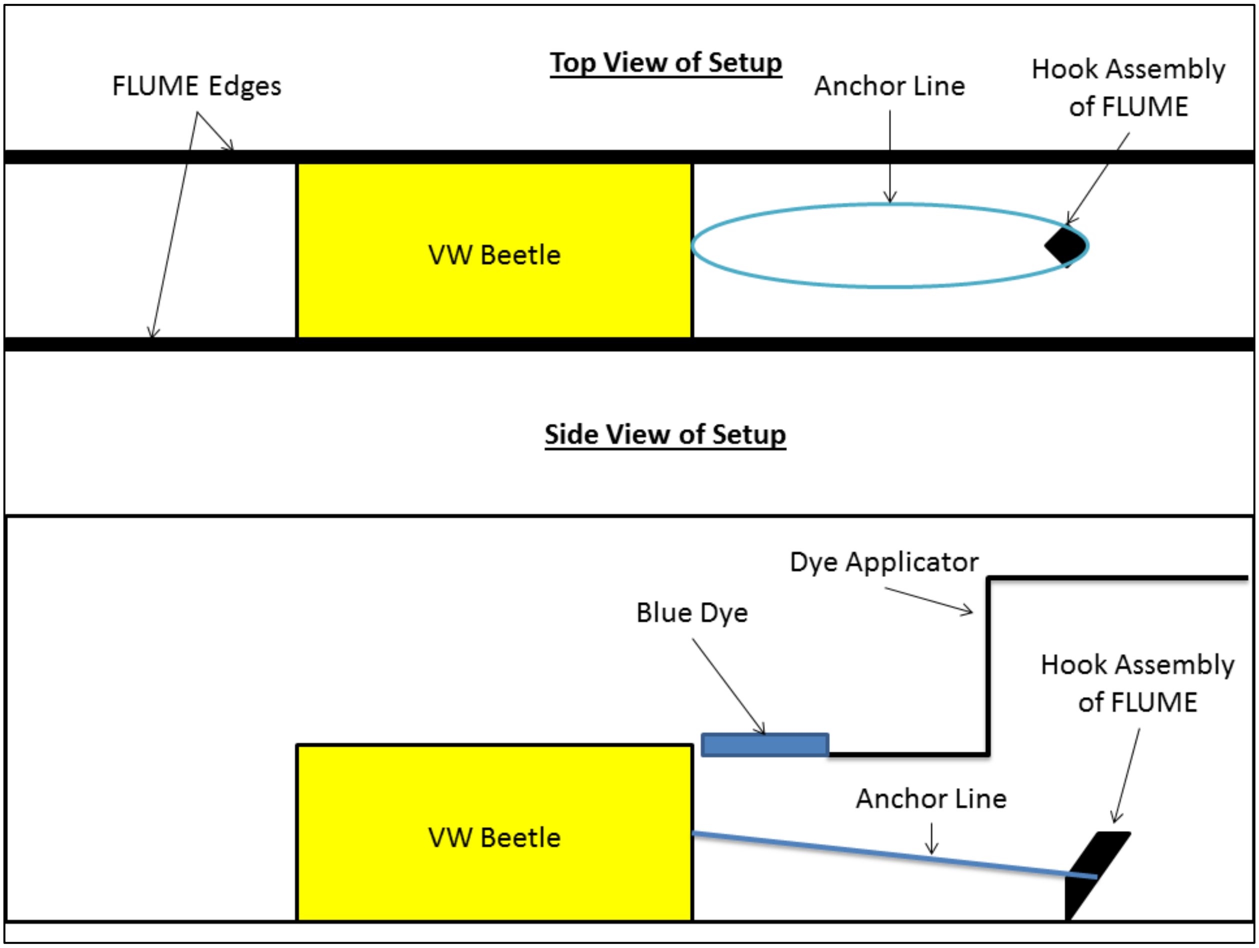

Figure 3-Top and Side View Schematic of FLUME setup for Team Photo #1

Note that in the following schematic, the light source was omitted in order to avoid clutter. As a general summary, a set light acquired from the ITLL was rested on top of the FLUME edges in order to create uniform lighting. In addition, the white background hooked to the back of the flume was omitted, but can be seen in Figure 1 (the background is slightly pink under the lighting). Note that the use of the light along with the white background created a pink background as opposed to a white background. While at first this was considered problematic, the pink hue provided a much sharper contrast between the fluid and the background. Additionally, this did not appear to be a problem beneath the surface of the fluid. When physically running the experiment, water acquired from a hose was used in order to supply the FLUME with the appropriate volume of fluid needed to perform this experiment.

Fluid Dynamics and Movement Simulation

Aerodynamics over a surface are used to optimize the performance of cars or airplanes. In general, the fluid dynamics for a car or airfoil for an airplane can be modeled by the Navier-Stokes Equations, as seen below in Figure 4 [1-6]:

$$\rho\left(\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{dy} + w\frac{\partial u}{\partial z}\right) =$$
$$\rho g_x - \frac{\partial p}{\partial x} + \frac{\partial}{\partial x}\left[2\mu\frac{\partial u}{\partial x} + \lambda\nabla\cdot\mathbf{V}\right] + \frac{\partial}{\partial y}\left[\mu\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right)\right] + \frac{\partial}{\partial z}\left[\mu\left(\frac{\partial w}{\partial x} + \frac{\partial u}{\partial z}\right)\right]$$

$$\rho\left(\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{dy} + w\frac{\partial v}{\partial z}\right) =$$
$$\rho g_y - \frac{\partial p}{\partial y} + \frac{\partial}{\partial y}\left[2\mu\frac{\partial v}{\partial y} + \lambda\nabla\cdot\mathbf{V}\right] + \frac{\partial}{\partial z}\left[\mu\left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y}\right)\right] + \frac{\partial}{\partial x}\left[\mu\left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x}\right)\right]$$

$$\rho\left(\frac{\partial w}{\partial t} + u\frac{\partial w}{\partial x} + v\frac{\partial w}{dy} + w\frac{\partial w}{\partial z}\right) =$$
$$\rho g_z - \frac{\partial p}{\partial z} + \frac{\partial}{\partial z}\left[2\mu\frac{\partial w}{\partial z} + \lambda\nabla\cdot\mathbf{V}\right] + \frac{\partial}{\partial x}\left[\mu\left(\frac{\partial w}{\partial x} + \frac{\partial u}{\partial z}\right)\right] + \frac{\partial}{\partial y}\left[\mu\left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y}\right)\right]$$

Figure 4-Navier-Stokes Equations used for characterization of 3-D flow over a surface

Note however that these equations are incredibly complex and can be simplified only when analyzing near ideal situations. Unfortunately, this is generally not the case. Additionally, the Reynolds-Average Navier Stokes is used more often than the full Navier-Stokes equations to form a reference grid over the object in question [2, 3, 6]. Consequently, computational fluid dynamics (hereafter referred to as CFD) can be used to characterize the flow. Additionally, due to advancements in computer technology, CFD analysis for flows with “nonlinear, rotational, viscous physics of transonic flows” [1, 4, 6] permit the analysis and further optimization of airflows over vehicles and planes. For the purposes of this experiment and due to its complexity, CFD was not used for flow analysis. Instead, using the scale of the model, which was 1:43 and an assumed speed of 20 mph it was possible to create flow simulations that would approximate the streamlines observed when driving a full-scale VW beetle at 20 mph. Some assumptions that were used for this fluid flow were that the flow was incompressible, the flow was relatively steady, and that internal flow within the model itself was negligible [3, 6]. Consequently, a flow rate of 0.1999 inch/sec was calculated which resulted in a Reynold’s Number of 9.3194, suggesting non-turbulent flow [3]. However, this was in contrast to observation as in Figure 1, the dispersal of the dye was characteristic of turbulent flow that would normally be observed. Through examination of the left side of the photograph, we saw that the wake flow behavior behind the body conformed to the shape of the car from the surface to the car trunk, which was to be expected for flows at velocities of 0.1999 inch/sec [4, 6]. As seen from Figure 1, the dye appeared to exhibit laminar behavior towards the back end of the car. In contrast, when the flow closer towards the windshield of the car was examined, extreme turbulent behavior was observed, specifically during dye impact with the windshield. This phenomenon was a result of the shear forces originating from the hood of the car and the faster flow over the top of the car, creating a rotational motion for the dye directly in front of the windshield [4]. Furthermore, there existed a distinct pressure difference between the top of the car and the area at the windshield. If we examine similar experiments regarding pressure distributions over Sedan car surfaces or hatchback car surfaces, there is a distinct difference in the magnitude of pressure at these locations [5, 6]. Figure 5 below indicates these differences [5]:

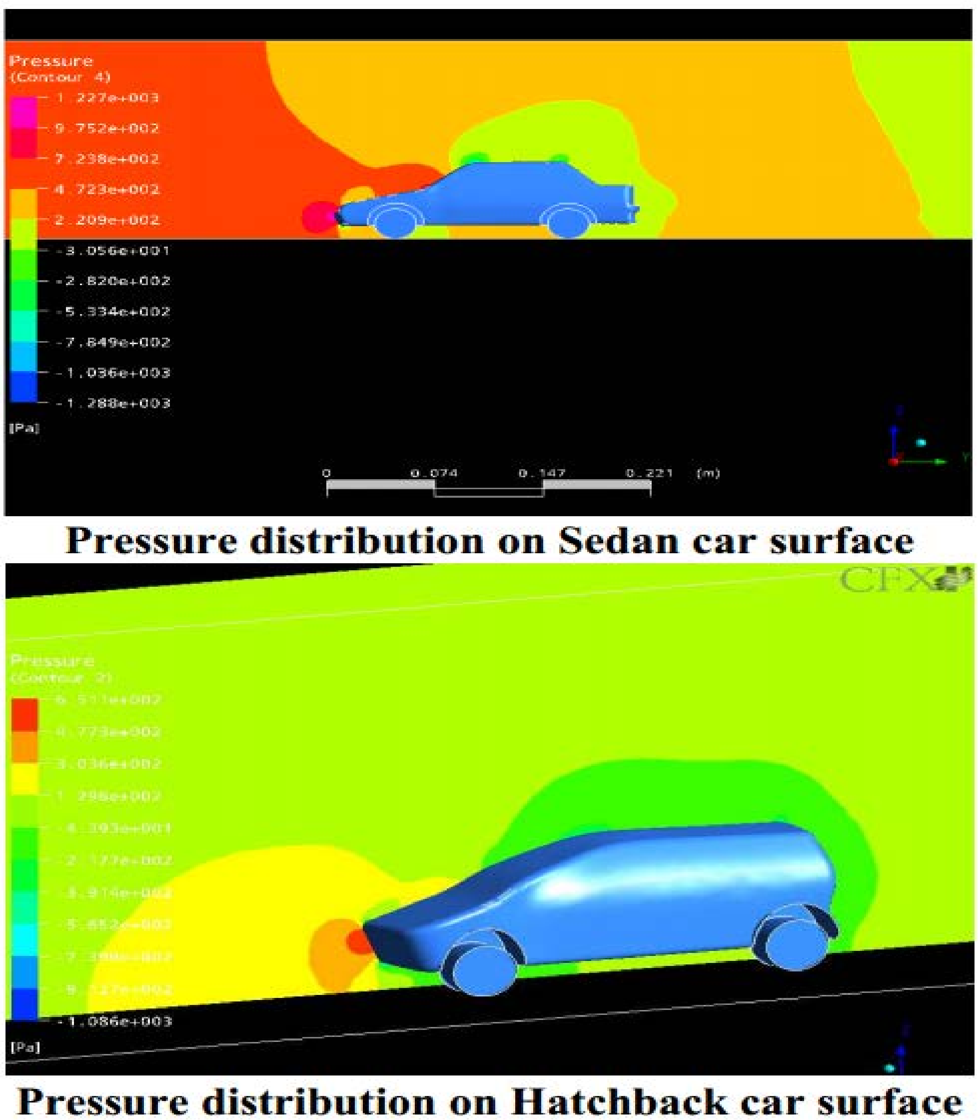

Figure 5-Pressure Distributions over two different car models from an Aerodynamics Performance Assessment in the Journal of Engineering Research and Studies

These differences in pressure distributions resulted in some mixing behavior due to a higher-pressure region of fluid attempting to collapse a lower pressure region [5, 6]. This too contributed to the turbulent behavior observed at the windshield location. Consequently, while the flow appeared to be turbulent, it was odd that the flow over the VW beetle was at turbulent conditions simply due to a low Reynolds Number. This will be addressed in more detail below. Some other interesting characteristics to note regarding the flow patterns observed over the VW Beetle were the vortex patterns towards the back of the car. It is difficult to visualize, but there is evidence a vortex is forming at the very end of the dye. When a car is moving at normal speeds, a flow field with this spinning motion can be observed through CFD or mass dye injection [4,6]. However, it is important to note that this turbulent behavior was not as apparent, nor expected due to the low Reynolds Number of the system. In this sense, as stated above, it was odd to see evidence of turbulent flow when the system itself had a Reynolds Number significantly lower than that needed for the transition from laminar to turbulent flow. Despite this fact, it is important to note that this was a scaled model. Under the conditions simulated for a 20 mph speed, it was likely that despite the lower speed of the fluid flow, turbulent behavior was still reflected even though the Reynolds Number was so low. This is primarily because scaling such flows over a model required a scaling of the Reynolds Number needed for a transition from laminar to turbulent flow. Consequently, the transition for this experiment would not be at 2000 as it would for a full-scale car, but at a significantly lower number that would need to be verified using CFD [1, 6].

Visualization Techniques

In order to visualize the flows seen in Figure 1, a water medium was chosen due to its consistency as well as its clarity within the FLUME. Additionally, the use of a wind tunnel was inadvisable due to the lack of anchoring surfaces. In order to ensure that the flow was clear, the FLUME's anchoring points were reconfigured to avoid any obstructing objects like tape or measurement devices that were attached to the FLUME tank wall. The FLUME was also configured with marbles in order to achieve smooth flow within the FLUME itself. In addition to ensuring smooth flow, the FLUME's marbles also eliminated a majority of the bubble content within the fluid itself, permitting clear flow patterns that did not contain any distracting elements. To capture the photo once the experiment was running, a tripod was set up in front of the VW Model's position such that the photographs were stable. To provide a consistent background, an attachable acrylic background was hooked to the edge of the FLUME and positioned behind the VW beetle such that no distracting background elements were present in the photograph.

To light this photograph, a stage light found in the ITLL was used to provide uniform lighting. Although originally, an apparatus was considered to mount the light on, it was found that simply using the FLUME's edges to rest the light upon worked more effectively for the purposes of uniform lighting. Additionally, the light used was susceptible to rapid temperature increases. Consequently, using the FLUME's edges eliminated the possibility of holding an extremely hot stage light. This method provided surprisingly clear photos, which enabled the capture of very subtle details within the photo used for the analysis of the physics, as seen in the previous section.

Photographic Technique

The image in Figure 1 was captured using a Canon EOS Rebel T5i. As mentioned earlier, a tripod setup was used to stabilize the photograph. In addition, since the FLUME was run continuously with clear water in preparation for the shot, the tripod served the dual purpose of examining different framing perspectives as well as stabilization. In order to avoid contact between the stage light and the water, a water level of 5.2" was used. This submersed the VW model car, preserved the flow patterns, and prevented any contact between the light and the water. To capture the photo, an estimated distance of 2.5" was present between the camera and the actual VW model itself. The height and width of the photo was 6" by 7". During the photoshoot, the focal length was set to approximately 28 mm with an aperture of 3.75. Note that because the lighting provided by the stage light was relatively uniform, the photo was taken with a "forced flash off" setting. Additionally, exposure time was 1/1250 sec with an ISO setting of 3200. Note that due to the relatively smooth flow of the water during the experimentation as well as the uniform texture of the water, motion blur was not a significant problem for the taking of this photo. The field of view was chosen specifically to mimic the classic "flow over an airfoil" photographs that are normally seen in textbooks. It was important to note that out of all the angles attempted, this provided the best vantage point to capture the flow behavior over the body of the car. In order to capture as much detail of the flow as possible, the exposure was chosen such that as much of the dye movement over the top of the car would be captured. Additionally, due to the bluish hue of the base of the FLUME, a chosen higher ISO setting served to highlight the contrast between the dye and the base, as well as eliminate some of the texture and scratches present upon the glass. Dimensions for the modified photograph were 700 x 411 pixels. Dimensions for the original photograph were 5184 x 3046 pixels. The original image is provided below as seen in Figure 6:

Figure 6-Original Unedited Photo detailing Fluid Flows Over a VW Beetle in a FLUME

The photograph was post processed in Adobe Photoshop CS5. General corrections included a cropping of background details in order to create a focus on the surface of the fluid as well as the VW model. Due to the high degree of lighting used to create the photograph, there were certain imperfections on the FLUME's glass that were detected. Consequently, the pink hue color of the background was identified in Photoshop using the dropper tool, and then used to cover all areas of the photo above the surface of the water. Contrast was not changed for this photograph as any modifications resulted in a darkening of the distinction between the dye and car body, resulting in the loss of certain turbulent characteristics associated with the flow pattern. Figure 7 is the modified version of the photo:

Figure 7-Edited Photo detailing Fluid Flows Over a VW Beetle in a FLUME

Overall Analysis

As a general summary, the following experiment utilized simulated conditions of a 20 mph VW Beetle to produce turbulent flows over the top of the model car. Dye was used to emphasize the streamlines and resolve the flow pattern behavior. From this emphasis, it was determine that low pressure and high-pressure regions appeared in the front of the windshield and on top of the car respectively. These coupled with shear forces at these regions and the turbulent behavior associated with a scaled Reynolds Number provided the jagged pattern of dye as seen at the windshield. Additionally, the wake flow behavior observed by the dye towards the end of the car in addition to the surface of the flow provided an opportunity for a clear visual analysis of the flow patterns towards the back, specifically the vortices generated by the turbulent flows running parallel to the VW's trunk. Overall, I am quite satisfied with the complexity of the physical flows that my team and I were able to reproduce for this project. More importantly, the use of the FLUME was very educational, not to mention quite fun. It was interesting to reproduce flow conditions needed to reach speeds of 20 mph, and more importantly, to draw comparisons between water as opposed to air when calculating flow conditions such as the Reynolds Number as well as flow. However, I found the dye dispersion to be somewhat limiting as a high concentration of dye was needed in order to create the flows that traversed over the body of the car. Additionally, it was difficult to resolve the flow lines over the body of the car as the dye created a sort of clumped appearance as it flowed over the car. I would have been interested in using different car models in order to see the different flow patterns. Unfortunately, more advanced cars were too big for the FLUME resulting in the inability to compare. Furthermore, it would have been interesting to use smoke lines as opposed to fluid to examine the flows, similar to those observed when testing airfoils. As an alternative to dye, I would have liked to mix in some kind of hydrophobic liquid such that two separate layers could be observed in the FLUME. Unfortunately, the use of any other fluid in the FLUME besides water is not recommended as it may cause damage to the machine. Overall, I am very satisfied with the photo, and would very much like to continue this project, but with different cars, shapes, or maybe even a different flow simulation machine all together.

References

[1] Eyi, S., J.O. Hager, and K.D. Lee. "Airfoil Design Optimization Using the Navier-Stokes Equations."*Journal of Optimization Theory and Applications*. 83.3 (1994): 447-461. Web. 20 Mar. 2014. <http://link.springer.com/article/10.1007/BF02207637>.

[2] Djojodihardjo, H., M.F. Abdul Hamid, et al. "Computational Study on the Aerodynamic Performance of Wind Turbine Airfoil Fitted with Coandă Jet." *Journal of Renewable Energy*. 2013. (2013): n. page. Web. 20 Mar. 2014. <http://www.hindawi.com/journals/jre/2013/839319/>.

[3] Ahmad, Nor Elyana, Essam Abo-Serie, et al. "Mesh Optimization for Ground Vehicle Aerodynamics."*CFD Letters*. 2.1 (2010): n. page. Print. <http://www.issres.net/journal/index.php/cfdl/article/viewFile/63/43Ahmad>.

[4] Dumas, Laurent. "CFD-based Optimization for Automotive Aerodynamics." *French National Centre for Scientific Research*. Universit´e Pierre et Marie Curie. Web. 20 Mar 2014. <http://dumas.perso.math.cnrs.fr/Cha1.pdf>.

[5] Zala, Bhagirath, Dr. Pravin P. Rathod, et al. "AERODYNAMIC PERFORMANCE ASSESSMENT OF SEDAN AND HATCHBACK CAR BY EXPERIMENTAL METHOD AND SIMULATION BY COMPUTATIONAL FLUID DYNAMICS- A REVIEW." *Journal of Engineering Research and Studies* . 3.1 (2012): 91-95. Web. 20 Mar. 2014. <http://www.technicaljournalsonline.com/jers/VOL III/JERS VOL III ISSUE I JANUARY MARCH 2012/Article 19 JERS Vol III Issue I.pdf>.

[6] Islam, M. M., and M. Mamun. "COMPUTATIONAL DRAG ANALYSIS OVER A CAR BODY ."*Proceedings of MARTEC 2010 The International Conference on Marine Technolog*. (2010): n. page. Web. 20 Mar. 2014. <http://www.buet.ac.bd/name/martec2010/2010/File23MARTEC182.pdf>.