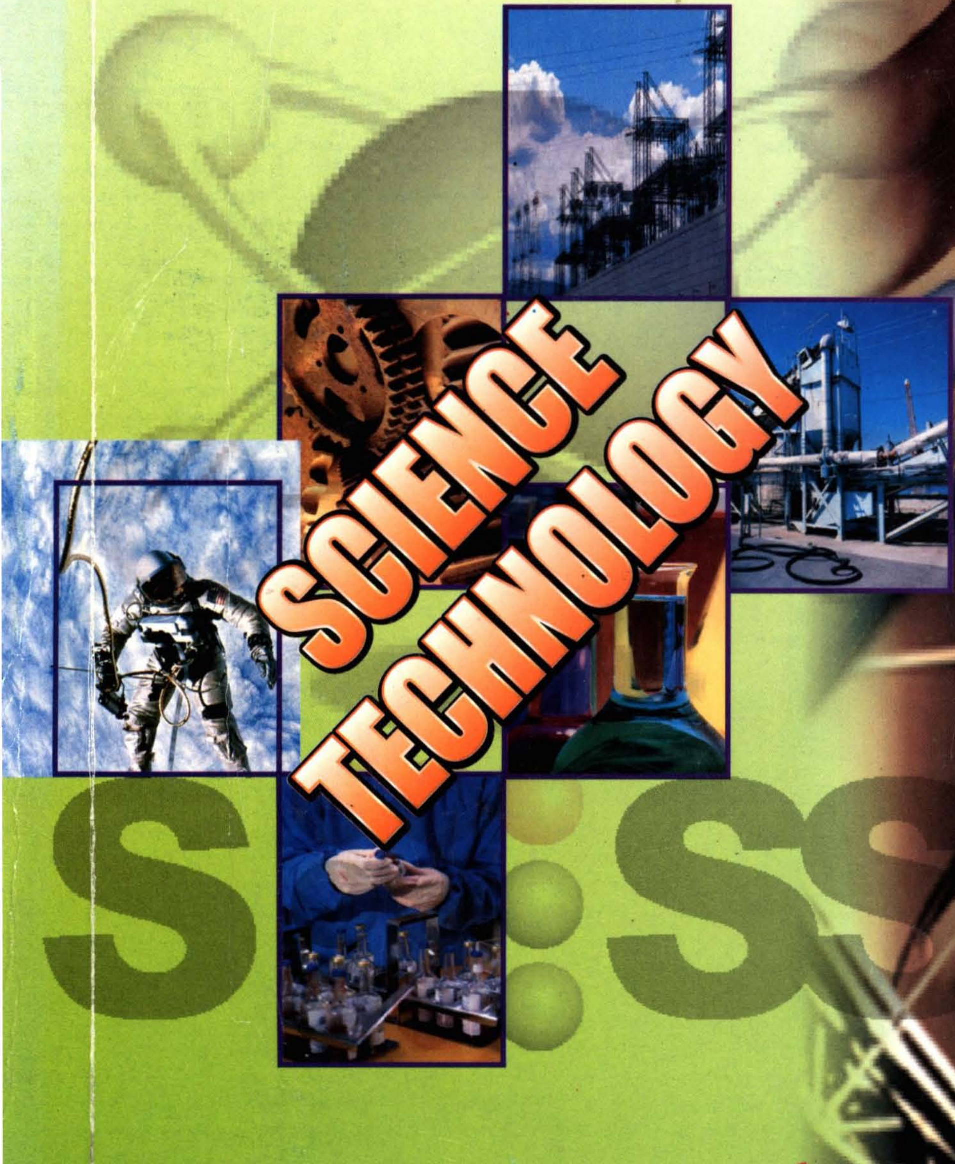
UNIVERSITI TEKNOLOGI MARA

ISO 9001:2000 No Sijil :0500132

Globalising Knowledge and Information

NATIONAL SEMINAR ON SCIENCE TECHNOLOGY & SOCIAL SCIENCES 2006



30-31 May 2006

Swiss Garden Resort & Spa

Kuantan, Pahang

CFD Analysis on Passenger Car Model for Drag Reduction

Lim Chee Hong
M. Govardhan
Mohammed Kamel Wan Ibrahim

ABSTRACT

The aerodynamic drag experienced by a vehicle is related to the structure of the flow in its wake. Hence experimental and computational studies are initiated on a typical passenger car. Wind tunnel testing and CFD analysis was performed on a 1/6 scale model of a Mercedes-Benz C45 model complete with 77 pressure tappings on the centerline contour. Accuracy of the model was preserved with fine details such as mirrors, bumpers, sidestrips and wheel housing. The vehicle yaw angle was varied from +15° to -15° and was shown to have a strong influence on the flow characteristics. Flow visualization via wool tuft technique at various yaw angles enabled the visualization of vortex formation along the A-posts and turbulence with flow separation at the rear end of the vehicle. Drag force was measured using experimental methods which yield drag coefficient values close to manufacturer claims. Extensive pressure measurements along the centerline including the front and rear diffuser area showed change in stagnation areas as the yaw angle of the car was increased. Data indicated well attached flow up to the top of the front screen. Stagnation points were visible at the front bumper area with flow separation starting at the end of the roofline. Experimental data was then compared with CFD software (COSMOS-FLOWORKS 2004) utilizing time dependent Navier-Stokes equations. The pressure distribution on the car surface correlated well with wind tunnel data.

Keywords: *Aerodynamics, CFD, drag coefficient, flow separation, wind tunnel test and pressure coefficient.*

Introduction

The reduction in drag in passenger cars goes a long way in increasing the fuel efficiency of the car. Studies in ways of reducing such problems may provide car aerodynamicists and designers alike, new approaches and guide to more effective car design. Although currently there are extensive aerodynamic studies on airplanes and aerodynamic research of automobiles, passenger cars are still at their infancy. This may be due to the large variations in passenger car designs and the current preferences of appearance or cosmetic beauty rather than aerodynamic considerations.

This project provides quantitative and qualitative results on drag reduction aspects of a typical 4 door sedan car. Quantitative tests include center-line pressure distribution, drag force and drag coefficient determination. Qualitative tests include Computational Fluid Dynamics (CFD) analysis and wool tuft flow visualization. The model being tested is a 1/6 model of a popular luxury sedan, which is known as the 1998 Mercedes Benz C45 (model W202). Analysis of the experiments will yield suitable aerodynamic enhancements to reduce aerodynamic drag.

Vehicle Aerodynamics

Aerodynamic is a branch of science relating to how air will flow around a given obstacle. Road vehicle aerodynamics, in itself is relatively different from aerodynamics of aircrafts and planes. This is due to the complex 3-dimensional flow of air, and regions of separation which are dependent on the contours of the car and ambient testing parameters. At present, nearly all aerodynamic designs for road vehicles relies on a combination of experimental results, experiences and physical understandings of how air flows behaved. Much aerodynamic development involves trial and error experiments using wind tunnel models (Barnard 2000). Scale models of the actual cars are mounted in the wind tunnel and tested for several parameters, which include, drag coefficient, lift and drag forces, pressure distribution, and flow visualization. However, recent advances in computation fluid dynamics (CFD) have enabled researchers to complement computer simulation with wind tunnel test results, thus enabling commercially viable aerodynamic breakthroughs.

The automobile drag force is dependent upon its shape, and is called as the aerodynamic resistance force (drag). Aerodynamic drag is the resistive force of air, which increases as the cube of relative speed of the vehicle and air. Drag depends on the vehicle's frontal area, shape, and body surface smoothness, and is measured by the drag coefficient (C_D), which is the non-dimensional ratio of drag force to the dynamic pressure of the wind on an equivalent area. This force is generated by two sources, which is air flow over the body and air flow over the radiator system as well as interior of the car. However air flow over the body or contour of the car accounts for more than 90% of the total aerodynamic force of a passenger car (Wong 2001). Aerodynamic drag starts to dominate typically at speeds around 60 km/h (Barnard 2000). Hence fuel consumption is reduced if drag coefficient values are lowered. Similarly acceleration can be improved by improving drag coefficient, which is characterized by Newton's Law of Motion for an accelerating vehicle.

The pressure over the vehicle varies across the surface and is dependent on the geometry of the vehicle. The pressure on the vehicle acts normal to the surface and contributes to the lift and drag forces accordingly. The pressure at each point on the surface of the vehicle can be characterized by the pressure coefficient (C_P) (Barnard 2000) The value of C_P is one (unity) at a stagnation point and is zero when the local and free-stream velocities are the same such as over flat sections of the vehicle. In regions of accelerated flow the pressure coefficient is negative. By knowing the value of pressure coefficient at that particular point, it is easy to calculate the free stream speed using the above relationship. Normally C_P values are attained at the centerline contour of the car model, and it has been shown that wind tunnel test experiments correlate well with CFD data as tested by Ferrari on its development of Ferrari Enzo (Buresti 2004).

Investigations of flow field around a vehicle lend much to the description of flow separation. During separation, the boundary layer flow no longer follows the car contour and separates away. Moreover, the pressure in the separation region is nearly equal to the pressure where separation of flow starts to take place (Janna 1993). Hucho (1987) pointed out that the wake or flow separation regions are strongly dependent upon the rear end shape of the car. He also pointed out that the wakes consists of quasi 2-dimensional wakes and longitudinal vortices. Ahmed (1980) in his experimental study of wake structure of typical automobile shapes concluded that the vortex system produced by altering the rear section slant angle design will give different drag coefficient values.

The aerodynamic drag comprises of two components, the skin friction drag and the pressure drag. Pressure drag arises from the component of normal pressure on the vehicle body acting against the motion of vehicle, while skin friction drag arises from the shear stress in the boundary layer adjacent to the external surface of the car (Wong 2001). Fluid mechanics state that when separation takes place, the amount of boundary layer normal pressure drag produced depends largely on where the flow separation occurs. A circular plate held normal to the flow will produce separation around the high periphery of the leading face, resulting in wide wake with high drag coefficient. Conversely, a teardrop shape with a long tail will retain attached flow to the end, with a consequently low C_D value (Douglas 2001). Hence, the two main criteria influencing bluff body aerodynamics is the roundness of its front corners and the degree of taper at its rear end

Drag Reduction Techniques

Drag reduction depends upon the reduction of friction, pressure, trailing vortex and excrescence drag. Skin friction can be reduced by designing a continuous smooth surface with no sudden changes in direction, gaps or surface detail. Pressure drag can be minimized by keeping attached flow as far back as possible, which implies continuous surface contours without facets or sharp corners. (Barnard 2000). In addition, pressure should be allowed to rise as much as possible towards the rear of the vehicle, or the cross sectional area should be gradually decreased towards the rear, such as a teardrop shape.

The easiest drag reduction method for streamlining the front end is to avoid any sharp corner above the radiator and a flat front. Drag reductions up to 14 % is achievable by modest rounding and lowering of this corner (Hucho 1987). Much more improvement is possible when the front end is made as a smooth continuous curve originating from the front bumper (Barnard 2000). Onorato et al. (1987) showed the effect of streamlining the rear end of a car to a taper akin to an aerofoil end would undoubtedly reduce the drag coefficient to a minimum.

The rear angle inclination known as the rake angle initially will cause C_D to fall with increasing rake angle (Barnard 2000). This is due to the decrease of pressure drag. However at 10°, the drag starts to rise due to formation of strong conical vortices. At angles greater than 30°, the vortices cannot form and separation occurs. (Barnard 2000). Nouzawa et al. (1992) concluded the same remarks after analyzing the aerodynamic drag of a notchback (sedan) model.

The underbody of a vehicle of floor pan is home to the exhaust pipes and fuel tank reservoir. For low drag design, the underbody should be as smooth and as flat as possible, to reduce unwanted flow separations and reduce surface friction drag. The addition of a diffuser, or an upsweep at the end of the underside would reduce the pressure form drag and reduces lift significantly (Barnard 2000). However, the inclination angle should be less than 15° so that moderate drag reduction effect is produced (Katz 2002).

Zhang & Ruhrmann (2003) in their investigation of diffuser angle on a bluff body in ground effect found that decreasing ride height and for low diffuser angles, the downforce is high due to the presence of two counter rotating vortices that prevent separation bubbles to occur. They surmised further that flow, through low angled diffusers is influenced by the underbody and presumably ground, boundary layers. Air dams or commonly known as front spoilers are bolt on devices used to restrict the flow of air from the front bumper to the underbody. According to the

Bernoulli relationship, such action would result in pressure reduction on the underbody, hence reducing lift and trailing vortex drag. The air dam would effectively lower the front stagnation line, which is favorable to drag reduction. However air dams only work if properly designed and have undergone wind tunnel test (Barnard 2000). Proprietary add-ons are more of fashion and were not always effective. Spoilers are upturns at the end of the boot (trunk lid). Their effect is the same as raising the effective boot height, resulting in lower rear end lift. Rear end spoilers however, are effective at reducing lift in the sacrifice of some increase of pressure drag. Adding spoiler height would decrease lift, but drag is increased as well (Fukuda et al 1994).

Experimental Setup

An accurate wood 1/6 scale model of the Mercedes-Benz C45 is used in this present investigation. The model is painted black to reduce radiation emmissivity and was bored with 77 pressure tappings. To avoid flow measurement discrepancies, the model is carefully carved with distinct areas such as bumper extensions, drip lines, mirror lines and windscreen area. The model has a defined wheel set complete with wheel housings and even tires have rounding on the sides. However from initial investigation, the wheels are locked and cannot be rotated. The underbody of the model is smooth and flat, thus reducing skin friction to a minimum. Hence drag reduction studies in this investigation concentrates on the upper body area.

77 static pressure holes each measuring 1 mm in diameter is bored on the centre-line contour of the model. The tappings include 4 pressure points at the bottom of the front diffuser and 3 pressure points at the rear diffuser. This enables full investigation on the flow of air even on the diffuser areas. The holes are brought down to a centralized centre tapping, where pressure tubes can be connected to a manometer for pressure readings. The pressure tappings are indicated in Figure1.

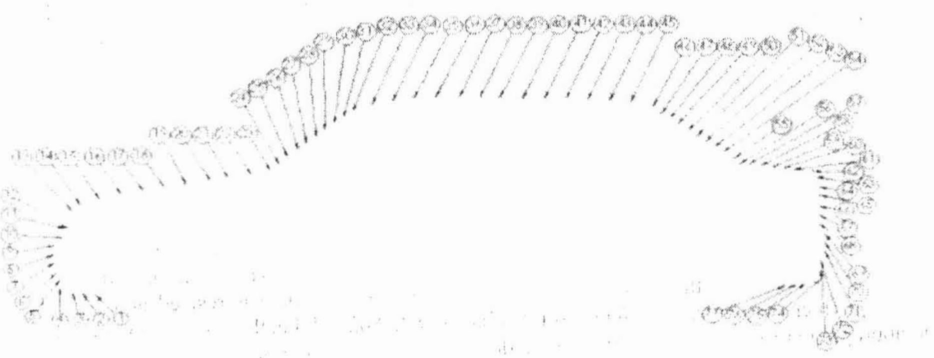


Fig. 1: The 77 Pressure Tapping Locations on the 1/6 Car Model

All the experiments conducted in this investigation have been conducted on the SKTM Wind Tunnel Lab. The SKTM Open Loop Low Speed Wind Tunnel (OLWT-1000) is an open circuit type wind tunnel with speed capability up to 50 m/s (180 km/h). It is powered by a 75 hp 3-phase induction motor, which connects to a 10 blade fan via 3 ribbed v-belts. Test section area is measured as 1000 x 1000 x 2500 mm and the tunnel is 14.5 m in length and 4 m in height. 4 anti turbulence sheets are fixed at the tunnel opening. Fan speed control is achieved via a frequency controller at the side, which has a range value of 0 – 50. Wind tunnel speed however is determined from the multiple tube manometer readings, taking into account the temperature at the testing condition.

Wool tuft testing is done by attaching short strips of wool or other lightweight material to the car body and the model subsequently tested in the wind tunnel. Selection of wool is important, where the lightest material that does not inhibit flow is chosen. The wools are cut into strands of 35 mm each, representing streamlines and hence representing flow in the region. Since the color of the model is black, contrasting colored wool is needed. Hence a light blue colored wool tuft is chosen for the experiment. Lightweight adhesive which is normal glue is used to stick the wool to the surface of the model. The normal glue is easily removed by water and has enough bonding strength for wind tunnel testing. Furthermore it does not damage the surface black coat of the model. The base of the wool is applied to the adhesive and thus only the base sticks whereby the remainder of the wool is free to move or oscillate. A total of 275 wool tufts are glued onto the surface of the model, with each wool having ample space for motion. Placement of tufts is critical to determine separation regions and turbulent areas. The model is left alone for one day for the adhesive to bond the wool and the surface of the car model.

The car model is mounted in the wind tunnel with 4 screw mountings and a base mount. This is to elevate the model enough so that boundary layer effect is minimized. The wind tunnel is started and the wool tufts are photographed at various angles, and the experiment is repeated for three speed settings and yaw angles of -15° to 15°.

Qualitative analysis is then performed by analyzing the photographs of the wool under testing conditions.

Drag Force testing was done by utilizing spring scale and mounting the car model on movable bearings with tracks for path adjustment. Since the model wheels are not movable, two shafts are machined out of mild steel and the bearings are subsequently glued using Loc-Tite solution onto the shaft. Two wooden blocks act as the base support and the shafts are mounted onto it via semi circular couplings and then nailed tight. 4 type 6301 non-covered ball bearings are used and it is cleaned with oil before being used to ensure low friction and free spinning is achievable. For the tracks, plastic wiring sheaths were used, and these were mounted on the wind tunnel test chamber floor. A small pulley was used and this connects the spring scale and the car model. The spring scale used is a 10 N spring scale with smallest increment at 0.25 N. The experiment is based on static equilibrium for objects in motion. Drag force induced on the car is measured by the spring scale. Here the spring scale measures the drag force corresponding to a certain wind tunnel speed. The car is pushed back, and force equilibrium will cause the car to be stationary again at a distance corresponding to the drag force applied. Hence here the drag force is obtained. Cross sectional area has been measured using CMM machine, which is 0.0496 m^2, with temperature constant at 30° C and atmospheric pressure (101300 Pa), density of air yields 1.165 kgm^{-3}. Velocity of the wind tunnel is taken from the difference in height of manometer rise. Blockage correction is calculated at 4.96 %, which is roughly acceptable, and the drag coefficient value is computed with the blockage effect taken into account. The experiment is performed at various speeds, or Reynolds Numbers, and the experiment is repeated three times. The drag coefficient values is computed and a graph of C_D versus Re is produced.

Static pressure test is conducted by connecting small clear plastic tubes to the bottom of the model, which then connects to the multiple water manometer unit. Since the static pressure hole are at the centerline of the model, the data obtained is the centerline pressure distribution, with some points defining the pressure at the front lower and rear diffuser.

The problem of connecting the manometer 1/8 inch (3.175 mm) clear plastic tubes to the 1/16 (1.58 mm) inch small tubing is solved by using hypodermic needles. The tips of the needles are grinded flat using the workshop grinding wheel. This is to prevent accidental tube piercing and damage. Next, the tubes are numbered accordingly, so that it is easily connected to the manometer unit. Then the model is mounted onto the wind tunnel. The experiment is conducted in the following method. Three definite speeds are chosen for testing. Then the model is mounted in the wind tunnel unit and at each speed, the model is tested at no yaw, 3, 6, 9, 12 and 15 degrees. Since pressure distribution is expected to be same for positive and negative yaw (centerline), positive values are tested only. The determination of pressure is calculated from the multiple manometer unit, and converted into C_P or pressure coefficient values.

Computational Setup

CFD testing is subsequently done to provide additional aerodynamic data to accompany the wind tunnel testing experiments. Here the CFD software used is the COSMOS-FLOWORKS 2004 from Dassault Systems. Using SOLIDWORKS 2004, a CAD model of the desired car is produced. The model is refined so that it takes the shape and intricate features of the actual model. The computational domain is resized and a total of 1,055,730 cells are generated which comprises of 128,003 fluid cells and 823,802 solid cells. The commercial RANS solver is then used and appropriate results are obtained. The model along with computational domain is shown in Fig.2.

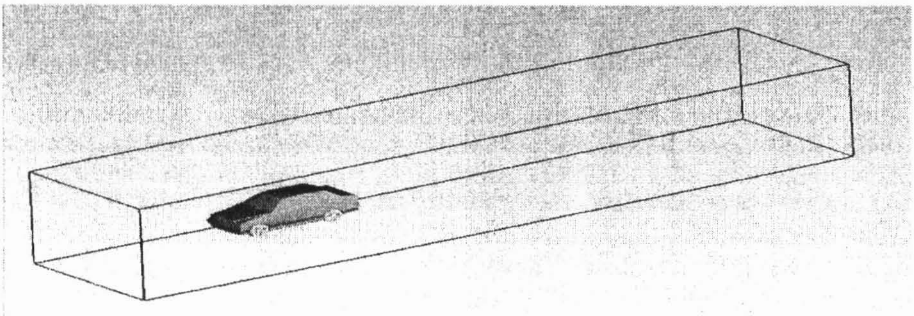


Fig. 2: The Computational Domain and the CAD Model Testing in COSMOS-FLOWORKS

Results and Discussion

Figure 3(a) shows the result of wool tuft testing for one speed setting, at wind speed of 18.35 ms^{-1}. Wool tufts that are straight and non-moving and oscillating slightly show attached flow which is laminar, and the flow adheres to the contour. Generally, equivalent positive and negative angles will have same flow characteristics, except that the flow is coming at a different direction. The flow is generally well attached up until the start of the windscreen, where the flow separates away, hence the tufts look disoriented. Formation of vortex can be seen for yaw angles of +9°, +12° and +15°, where the tuft try to separate away from the front side A-posts. Separation direction is dictated by free stream wind direction, as seen at the picture for angle -15°, (Fig. 3(b)) where at the start of the windscreen, the tuft direction points in the direction of air flow (The longitudinal axis of the car is yawed at -15°). Regions on the roofline show well attached flow until the back. Regions after the end of the roofline curvature are regions of turbulence and strong separation, due to the messy and strong oscillations of the tufts.

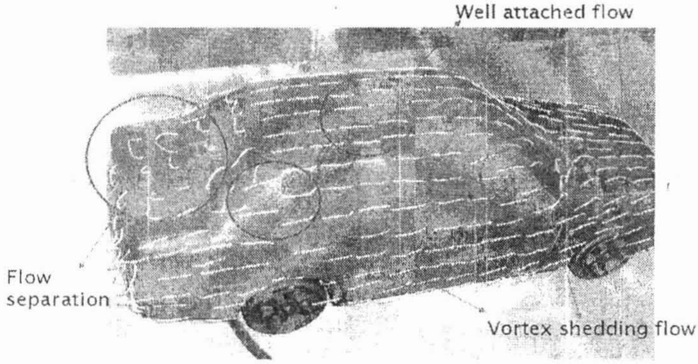


Fig. 3(a): Wool Tuft Testing at No Yaw with free stream speed of 18 ms^{-1}

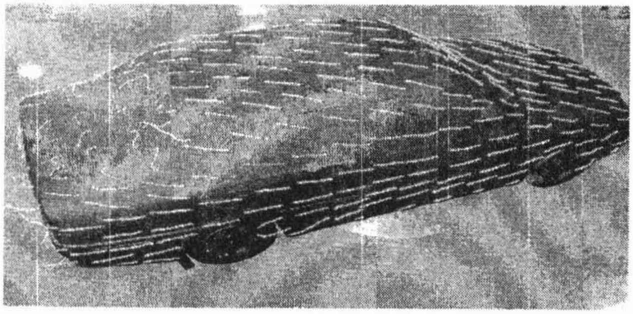


Fig. 3(b): Wool Tuft Testing at 15^0 with free stream speed of 18 ms^{-1}

Experimental drag force testing revealed values roughly around 0.37, which is slightly higher than the manufacturer claim at 0.32. Literature states that experimental data will yield higher C_D values, however further refinements can be made.

Figure 4 shows the wind tunnel pressure distribution test results. Generally at regions of high static pressure, as in stagnation area in front of the car near the bumpers, the C_P values are at unity. This indicates flow stagnation regions. As the air flow climbs the gradient up, and at the leading edge of the hood, the flow accelerates up, indicated by the drop in C_P value to a minimum. As the flow approaches the start of the windscreen, the flow is again inhibited, the stagnation areas arise again. Point 40 onwards denotes the rear windscreen area, where flow separation occurs, and the C_P values tend to average to similar values, as anticipated. Also, the onset of flow separation is denoted by C_P value near to zero, and is observed for different speed settings as well. As the yaw angle is increased, it can be seen that the corresponding C_P values decrease.

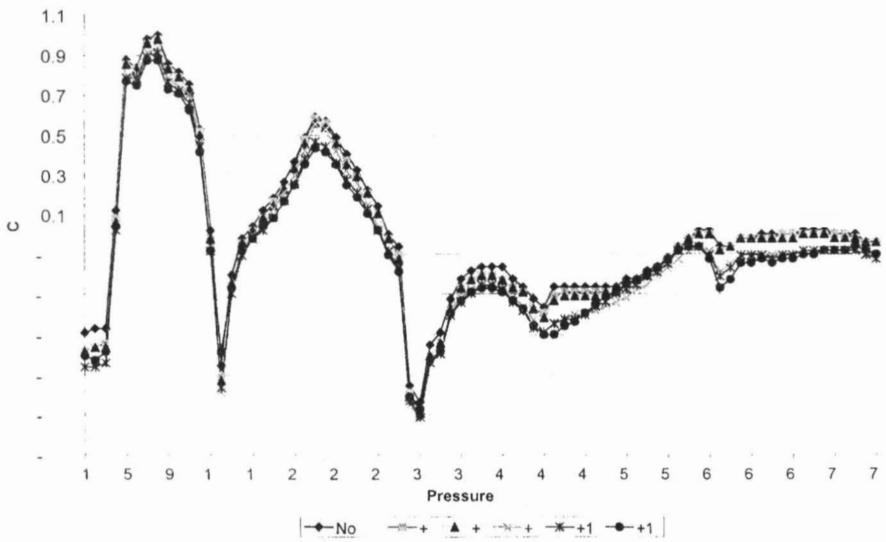


Fig. 4: The Centreline Pressure Distribution of the Car at Free Stream Speed 28 ms^{-1}

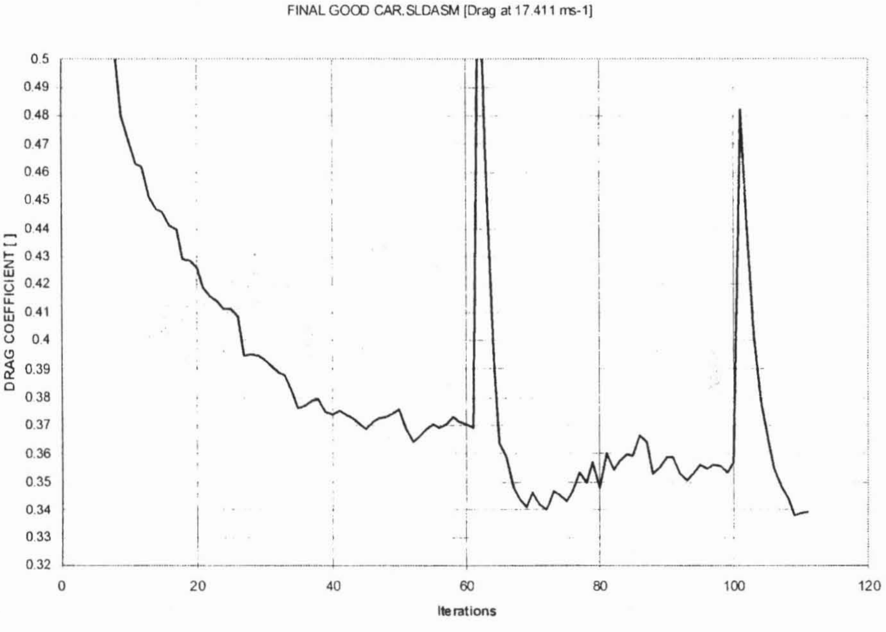


Fig. 5: The Drag Coefficient value Calculated via COSMOS-FLOWORKS.

Computational work performed involved using CFD analysis via COSMOS-FLOWORKS revealed drag coefficient data value at an average value of 0.359, with a total of 110 iterations. The computational work performed also shows strong correlation of the centerline pressure distribution of the car model using the same ambient characteristics.

Conclusion

This project has successfully investigated the aerodynamic characteristics of a common sedan passenger car. Wool Tuft Testing revealed attached flow, vortex shedding regions and separation regions on car contour. Pressure distribution test revealed quantitative results on stagnation region, flow separation region, and flow characteristics.

Computational CFD work performed agrees well with the experimental results.

References

Ahmed S.R. (1980). An Experimental Study On The Wake Structure Of Typical Automobile Shapes. *Proceedings of the 4th Colloquim on Industrial Aerodynamics, Fachhochschule Aachen.*

Barnard R.H. (2000). *Road Vehicle Aerodynamic Design: An Introduction.* London: Longman Publishing.

Buresti G. (2004). The Influence Of Aerodynamics On The Design Of High-Performance Road Vehicles - Part 2. *Research Paper. Department of Aerospace Engineering.* Italy: University of Pisa.

Douglas, J.F., Gasiorek M.J., and Swaffield J.A. (2001). *Fluid Mechanics.* 4th Edition. Essex.: Prentice Hall International.

Fukuda H., Yanagimoto K., China H., and Nakagawa K. (1994). Improvement of Vehicle Aerodynamics by Wake Control, *Japanese Society Of Automotive Engineers (JSAE).* 16: 151-155.

Hucho W.H. (1987). *Aerodynamics of Road Vehicles.* London: Butterworth Publishing.

Janna W.S. (1993). *Introduction to Fluid Mechanics.* 3rd Edition. New York: Longman Publishing.

Katz, J., (2002), *Race Car Aerodynamics.* Cambridge: Robert Bentley Publishing.

Nouzawa T., Hiasa K., Nakamura T., Kawamoto K. and Sato H. (1992). Unsteady Wake Analysis of the Aerodynamic Drag on a Notchback Model with critical afterbody geometry. *Society of Automotive Engineers (SAE)* SAE SP-908 paper 920202.

Onorato M., Costelli A.F., Garrone A., and Viassone L. (1987). Experimental Analysis of Vehicle Wakes. *Journal of Wind Engineering and Industrial Aerodynamics.* 22: 330-317.

Wong J.Y. (2001). *Theory Of Ground Vehicles.* 3rd Edition. New York: John Wiley & Sons.

Zhang Y., and Ruhrmann A. (2003). Influence of Diffuser Angle on A Bluff Body in Ground Effect. *Journal of Fluid Engineering, Transactions of ASME March 2003.* 125: 332- 338.

LIM CHEE HONG, M. GOVARDHAN & MOHAMMED KAMEL WAN IBRAHIM, Mechanical Engineering Programme, School of Engineering and Information Technology, Universiti Malaysia Sabah, 88999 Kota Kinabalu, Sabah. gova@ums.edu.my.