Applied Mechanics and Materials *Online: 2011-10-24*
ISSN: 1662-7482, Vols. 110-116, pp 2521-2528
doi:10.4028/www.scientific.net/AMM.110-116.2521

Active Flow Control over the Car

Deepesh Kumar Singh and Prof. Gautam Bandyopadhyay

Department of Aerospace Engineering

Indian Institute of Technology, IIT, Kharagpur (India)

deepesh.iitkgp@gmail.com, gbandyo@aero.iitkgp.ernet.in

Keywords- Active flow control, Turbulent flow, turbulent boundary-layer, suction/injection, flow over a car

Abstract. Active flow control methods are used to reduce the aerodynamic drag over a car model. Method of Boundary layer suction at the top rear and air injection at the back of the car are used as the active flow control tools to suppress the aerodynamic drag. The computational results obtained using the standard $k-\varepsilon$ model for the car model are verified first against the practical results obtained by wind tunnel experimentation so as to obtain the range of turbulence. Then a parametric study on the effect of the drag and lift coefficient of the car with respect to the parameters governing the active flow control is done. The drag coefficient is reduced by 20.25% using this strategy with 19.4% increase in the lift coefficient.

Introduction

Aerodynamic Drag and lift control over the automobiles is a matter of huge concern as it reduces the fuel efficiency and performance of the vehicle to a large extent. A considerable amount of research has been performed using passive methods of flow control which modify the flow around a vehicle without external energy expenditure [1]. Passive techniques include geometric shaping to manipulate the pressure gradient, the use of fixed mechanical vortex generators for separation control, and placement of longitudinal grooves or riblets [2] on a surface to reduce drag [3]. These methods have limitation and can not work beyond a point. In Active flow control, energy or auxiliary power is introduced into the flow with the intent to delay/advance transition, to suppress/enhance turbulence, or to prevent/promote separation [4]. The resulting benefits include drag reduction, lift enhancement, mixing augmentation, heat transfer enhancement and flow-induced noise suppression [5] [6] [7]. In this paper, the active flow control technique is been adopted to modify the flow over a car with the intent of reducing the aerodynamic drag. A car model present in the aerodynamic test facility of the Department of Aerospace Engineering, Indian Institute of Technology, IIT Kharagpur, India is taken into account. The geometry of the car model is shown in figure 1. First of all, 2-d k-epsilon steady flow calculations are done on this model at wind speed = 25m/s. These computational results are verified against the practical results obtained from the wind tunnel testing so as to validate the $k-\varepsilon$ model adopted for the modeling the active flow control. After this, the computational modeling of the boundary layer suction at the rear roof of the car and air ejection through the back of the car is employed and the value of lift and drag coefficient are monitored. All the calculations are performed using commercial software Fluent Inc.

Literature review

The passive Flow controls methods over a car to reduce the drag and lift coefficient include the judicious choice of bonnet slope and windscreen rake, roof and side panel cambering, underbody rear end upward taper, rear tail extension and the use of rear end spoiler.

A typical flow profile over a car body is shown in Fig. 3. A boundary layer over the forward surface of the roof will become laminar but further to the rear, a point will be reached called the transition point when the boundary layer changes from lamina to turbulent one. As the speed of the car raises further, the transition point moves further to the front and hence more turbulence in the flow. This boundary layer separation results the advancement in the drag. The space after the rear end of the car is occupied with the circulating air which is referred to as separation bubble or rear end suction which again causes drag augmentation.

Two prominent sources of Aerodynamic drag over the car defined by Heinz Heisler, Advanced Vehicle Technology [8] are a) flow separation over the rear –top section of the car and b) formation a pressure bubble behind the car, Fig. 1. In this paper, these two sources of drag are taken into focus and an active flow control techniques are adopted to prevent the boundary layer separation at the rear-top and formation of the pressure bubble at the read end of the car.

Actice Flow control technique adopted

In the two dimensional modeling of the car, 6 holes are modeled at the rear of the car top and 6 holes at back of the car referred as inlet and exit holes respectively. In 3-D, these holes will look like the rectangular holes (splices) as shown in Fig. 4. All the holes have the same width h and thus the net area of the inlet holes is equal to net area of the exit holes in 3-D. The distance between two consecutive holes is H = 0.02m. The boundary layer separation is prominent above the inlet holes and there is a pressure bubble in the vicinity of the exit holes.

A two step procedure is adopted to control the flow actively as follows:

a) The very weak boundary layer above the inlet holes is sucked in with a velocity v, Fig. 3 along negative y direction, i.e. $\alpha = 90^0$, Fig. 4. The basic principle of boundary suction is that the low-energy or air near the surface approaching the separation point is removed through a suction slot [9]. The result is a much thinner, more vigorous, boundary layer that is able to progress further along the surface against the adverse pressure gradient without separating, Fig. 4.

b) The mass of air being sucked in is injected with velocity v through the exit holes into the rear of the car where the effect of pressure bubble is prominent. The air is injected along positive x direction. Since flow separation is due to the complete loss of kinetic energy in the boundary layer immediately adjacent to the wall, a method of preventing it is to re-energize the tired air by blowing a thin, high-speed jet into it [10].

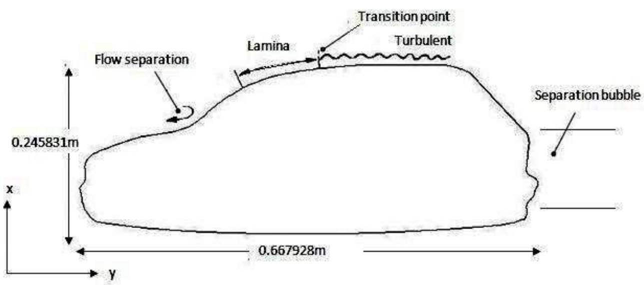

Figure 1:Geometry of the car model considered

Figure 2. Aerodynamics test facility, IIT Kharagpur, India.

A pressure pump is the driving unit which is assumed to suck and inject the air with the velocity ‘v’. The two step procedure viz. suction and injection has dual advantages. The boundary layer suction through the inlet holes causes the flow to remain attached to the car and hence decreases the drag. The air injection through the rear end decreased the trailing vortices and the extent of rear end suction or pressure bubble and hence contributes in drag reduction.

MATHEMATICAL MODELLING

The approach is based upon the turbulent boundary layer equations written in the form for 2-d problem

$$\frac{\partial \overline{u}}{\partial x}+\frac{\partial \overline{v}}{\partial y}=0 \quad ...(1)$$

$$\overline{u}\frac{\partial \overline{u}}{\partial x}+\overline{v}\frac{\partial \overline{u}}{\partial y}=-\frac{d\overline{p}}{dx}+\frac{\partial}{\partial y}(\mu\frac{\partial \overline{u}}{\partial y}-\rho\overline{u'v'}) \quad ...(2)$$

Where $\overline{u},\overline{v}$ and u',v' are the average and fluctuating components of velocity at any point in flow domain along x and y direction respectively. μ and ρ are air viscosity and density respectively.

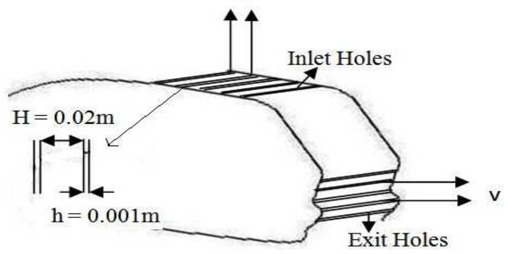

Figure 3: Geometry and specification of Inlet and Exit holes

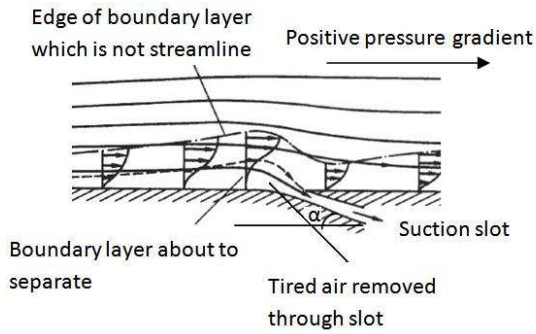

Figure 4: Boundary layer suction at an angle α

The basis of the $k-\varepsilon$ model used is an eddy viscosity formulae based on dimensional reasoning and taking the form

$$\varepsilon_T = C_\mu k^{1/2} l \quad ...(3)$$

Where C_μ is an empirical constant. The two equation models like zero-equation methods and Cebeci method generally have a transport equation based upon the length scale l. This seemed rather unphysical so, based upon the dimensional reasoning, the $k-\varepsilon$ model took

$$l = k^{3/2}/\varepsilon \quad ...(4)$$

Where k is the kinetic energy per unit mass of the fluctuating components i.e. $k=(u'^2+v'^2)$ and ε is the viscous dissipation rate per unit mass. A transport equation for ε can be derived from Navier-Stokes equations. Both the equations for ε and turbulent kinetic energy k contains terms involving additional unknown dependent variables. These terms must be modeled semi empirically. For our case, the transport equation for k and ε are modeled as follows:

Turbulence energy:

$$\rho\frac{Dk}{Dt}=\frac{\partial}{\partial y}\left(\frac{\rho\varepsilon_T}{\sigma_k}\frac{\partial k}{\partial y}\right)+\rho\varepsilon_T\left(\frac{\partial \overline{u}}{\partial y}\right)^2-\rho\varepsilon \quad ...(5)$$

Energy dissipation

$$\rho\frac{D\varepsilon}{Dt}=\frac{\partial}{\partial y}\left(\frac{\rho\varepsilon_T}{\sigma_\varepsilon}\frac{\partial\varepsilon}{\partial y}\right)+C_1\frac{\varepsilon}{k}\rho\varepsilon_T\left(\frac{\partial\overline{u}}{\partial y}\right)^2-C_2\frac{\rho\varepsilon^2}{k}\quad ...(6)$$

These equations contain 5 empirical values that are given the following values

C_μ	C_1	C_2	σ_k	σ_ε
0.09	1.44	1.92	1.0	1.3

Where σ_k and σ_ε are effective turbulence Prandtl numbers. Equation (5) and (6) are then discredited using the finite volume method. The initial guess for k is obtained from the turbulence intensity I of 10%

$$k=1.5(Iu_{avg})^2$$

Where u_{avg} is the average velocity over the flow domain. The initial guess for ε is made in a way so that the resulting eddy viscosity, $\varepsilon_T=C_\mu k^2/\varepsilon$ is sufficiently large compared to the molecular viscosity. In fully developed turbulence, the turbulent viscosity is roughly two orders of magnitude larger than the molecular viscosity and thus, the initial guess for ε is fixed.

GRID CONVERGENCE

Like the common practice in vehicle aerodynamics studies, a local refinement near the vehicle surface and in the areas of relatively high velocity gradients is performed. A mesh independency study is then carried out in order to ensure that the solution is independent of the mesh size. This is achieved through a further reduction in mesh size until a small difference in the drag coefficient between two successive mesh refinements is reached. This process continues until the difference in the drag coefficient is below 0.003 [11] or less than 5% [12]. The near wall distance is chosen so that the value of wall y^+ remains under 5.

RESULTS

The velocity of air encountered by the car is fixed at 25m/s (90km/hr) for the whole analysis. First of all the computations are done for the car model without inlet and outlet holes and results are verified with the wind tunnel testing at stage 3, where the air flow velocity at the car is equal to 25m/s. Shown in Fig. 5 is the value of pressure coefficient, Cp at different axial locations of the car top obtained from computational and experimental results. The maximum difference in the respective values of Cp was less than 5% and thus the k-epsilon model is verified. The value of lift coefficient Cl and drag coefficient Cd observed for the car model are tabulated in Table 1.

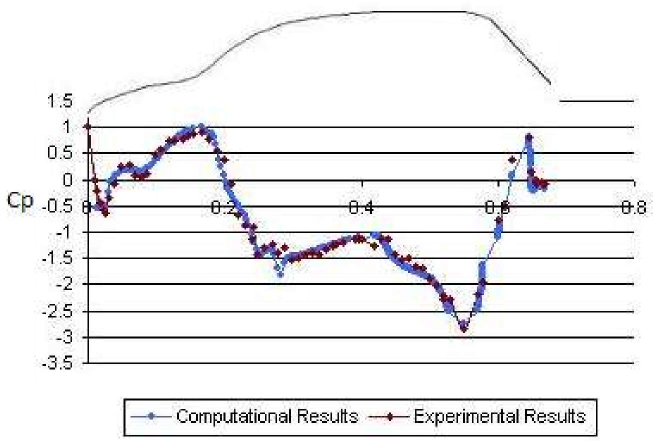

Figure 5: Comparison of pressure coefficient Cp values obtained from computational and experimental results over the top of the car.

TABLE I. COMPUTATIONAL AND EXPERIMENTAL VALUES OF DRAG AND LIFT COEFFICIENT FOR CAR WITHOUT INLET AND EXIT HOLES

	Drag Coefficient Cd	**Lift Coefficient Cl**
Computational	0.27837937	-0.9857
Experimental	0.279	-0.986

As mentioned earlier, the inlet holes suction velocity is equal to the exit holes injection velocity. This velocity v will be called as hole velocity only from now on. The car velocity is fixed at 25m/s as stated earlier and a parametric study of the effect of 'v' on Cl and Cd is necessary so as to know the range of values of hole velocity v for which the active flow control strategy is more effective.

A. Effect of hole velocit, *v* on drag coefficient *Cd*. As the hole velocity v is increased, the value of Cd decreases first and takes the least value at v = 60m/s. A further increase in v causes the Cd to increase tremendously and after v = 80m/s, the value of Cd saturates as shown in Fig. 5.

As can be seen from the table 2, the maximum drag reduction is obtained at v = 60 m/s which is 2.4 times the flow velocity over the car. Hence for the flow velocity of 25m/s, this method of drag reduction works best at the hole velocity v of magnitude 2.4 times the car or flow velocity. A maximum of 25% of drag reduction is achieved while operating at maximum efficiency (v = 60m/s) of the flow control device.

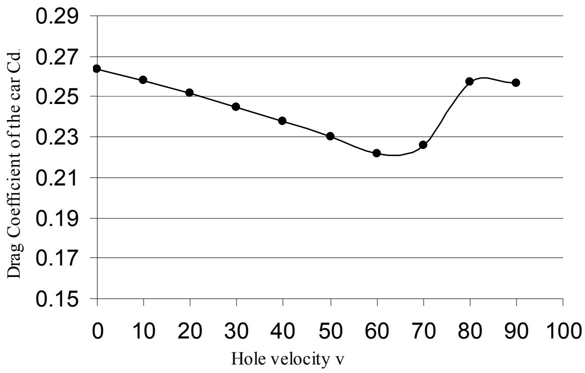

Figure 6: Drag coefficient of the car as a function of holes velocity

TABLE II. DRAG COEFFICIENT VALUES

hole velocity, v(m/s)	**Drag Coefficient Cd**	**Percentage reduction in Cd**
10	0.25804721	7.304
20	0.25153378	9.644
30	0.2447854	12.07
40	0.23783957	14.56
50	0.23008529	17.35
60	0.22201363	20.25
70	0.22600188	18.82
80	0.25759076	7.468
90	0.25639269	7.898

B. Effect of hole velocity *v* on lift coefficient *Cl*. As the hole velocity v is increased, the value of Cl increases steadily and saturates after v = 70m/s.

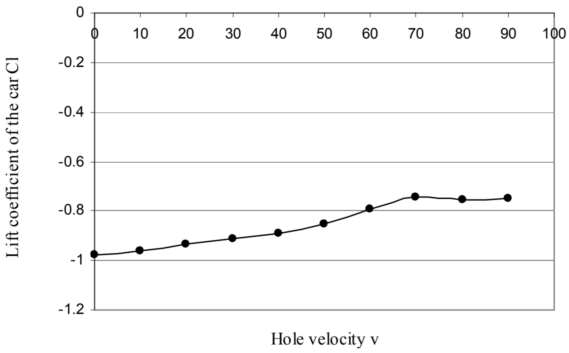

Figure 6: Lift coefficient of the car as a function of hole velocity v (m/s)

TABLE III. LIFT COEFFICIENT VALUES

Hole velocity, v(m/s)	Lift Coefficient Cl	Percentage increase in Cl
10	-0.9601471	2.622
20	-0.936619	5.01
30	-0.9136563	7.34
40	-0.8895819	9.78
50	-0.8549721	13.3
60	-0.794937	19.4
70	-0.7427269	24.7
80	-0.7573582	23.2
90	-0.7501503	23.9

While operating at minimum drag condition i.e. hole velocity v = 60m/s, the percentage increase in drag is 19.4%.

C. Boundary layer suction at an inclination. The boundary layer was been sucked at intake angle $\alpha = 90^0$ for the results shown above. A study of sucking the boundary layer at different values of intake angle α is done in this section.

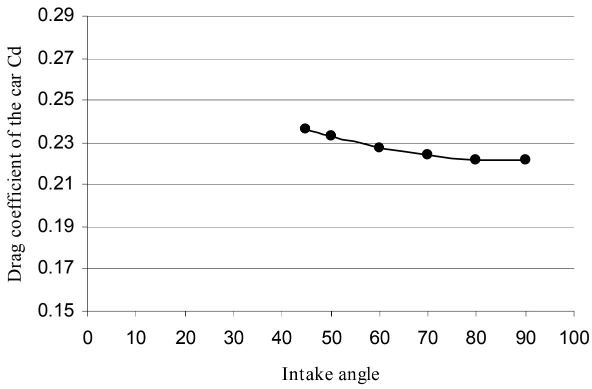

Figure 7. Drag coefficient Vs intake angle α

While reducing the value of α from 90^0 to 45^0 , a 10%increase in the value of drag coefficient Cd is observed, as shown in fig. 7 however, the lift coefficient Cl showed a huge deviation of 15% decrease Fig. 8. A decrease in the lift coefficient Cl is added advantage as it will increase the road holding of the tires. Hence, one can think of reducing the drag of the car by decreasing the intake angle α for the suction without compromising the lift.

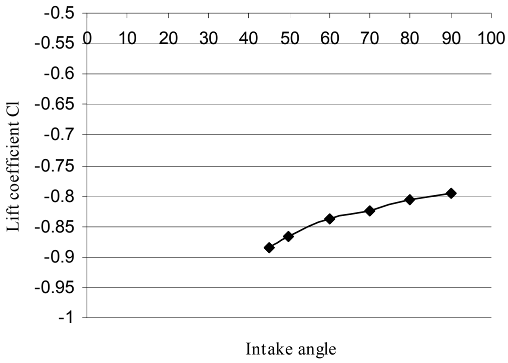

Figure 8. Lift coefficient Vs intake angle α (in degrees)

Comment on energy efficiency

For the car model considered, the gain of power developed by the drag force at 25m/s flow speed while operating at 60m/s hole velocity v is 19.37watts. The power consumed by a pressure pump to operate this device at 60m/s is 13.23watts and hence this method is energy efficient.

Conclusion

Boundary layer suction at the rear top of the car and air injection at the back of the car has been used as an active flow control tool to reduce the aerodynamic drag over a car. A 22.5% drag reduction is achieved using this method at 90km/hr car speed along with a lift penalty of 19%. In road vehicles like car, 19% increase in lift is not a major problem at 90km/hr speed however the lift penalty could be reduced to some extent by sucking the boundary layer at an inclination angle. This analysis is done for Reynolds number 427866. If the car speed is increased more, this strategy will count to be more effective because it will be able to control more turbulence in that case and hence more drag reduction.

References

[1] Bushnell, D. M. and Hefner, J. N., 1990, "Viscous Drag Reduction in Boundary Layers," Progress in Aeronautics and Astronautics, Vol. 123, AIAA, Washington, D.C.

[2] P.W. Carpenter (1990) 'Status of transition delay using compliant walls',

[3] Gad-el-Hak, M., Pollard, A., and Bonnet, J. P., 1998, "Flow Control: Fundamentals and Practices," Springer Lecture Notes in Physics, New Series Monographs, M53, Springer-Verlag, Berlin.

[4] Lumley, J. and Blossey, P., 1998, "Control of Turbulence," Annual Review of Fluid Mechanics, Vol. 310, pp. 311-327.

[5] Pack, L. G. and Seifert, A., 1999, "Periodic Excitation for Jet Vectoring and Enhanced Spreading," AIAA Paper No. 99-0672, 37th AIAA Aerospace Sciences Meeting, Reno, NV, January.

[6] Seifert, A., Bachar, T., Koss, D., Shepshelovich, M., and Wyganski, I., 1993, “Oscillatory Blowing: A Tool to Delay Boundary-Layer Separation,” AIAA Journal, Vol. 31, No. 11, pp. 2052-2060.

[7] Jonathan W. Furlong L. Ken Lauderbaugh Saunders Active Flow Control of a Bluff Body using FEMLAB with MATLAB

[8] Heinz Heisler, Advanced Vehicle Technology second edition, chapter 14, Butterworth-Heinemann publication house.

[9] L. Prandtl(l904) 'Uber Fliissigkeitsbewegung bei sehr kleiner Reibung', in Proc. 3rdZnt. Math. Mech., 5, 484-491, Heidelberg, Germany.

[10] P.W. Carpenter and P.N. Green (1997) 'The aeroacoustics and aerodynamics of highzspeed Coanda devices', J. Sound & vibration, 208(5)7,7 7-801

[11] Yang, Z., Schenkel, M., Assessment of Closed-Wall Wind Tunnel Blockage Using CFD. SAE Technical Paper 2004-01-0672, 2004.

[12] Murad, N., Naser, J., Alam, F., Watkins, S., Simulation of Vehicle A-Pillar Aerodynamics using Various Turbulence Models. SAE Technical Paper 2004-01-0231, 2004.