EXPERIMENTAL INVESTIGATION OF DRAG REDUCTION ON AHMED CAR MODEL USING A COMBINATION OF ACTIVE FLOW CONTROL METHODS

M. Jahanmiri

Department of Mechanical & Aerospace Engineering, Shiraz University of Technology, P.O.Box 71555-313, Shiraz, Iran
Depatrtment of Applied Mechanics, Chalmers University, Gothenburg, Sweden
Jahanmiri@sutech.ac.ir

M. Abbaspour*

Department of Mechanical & Aerospace Engineering, Shiraz University of Technology, P.O.Box 71555-313, Shiraz, Iran
m.abbaspour@sutech.ac.ir

*Corresponding Author

Abstract   Aerodynamic drag is an important factor in vehicle's fuel consumption. Pressure drag which is the main component of total drag is a result of boundary layer separation from vehicle surface. In this paper, we investigate experimentally, the effect of suction and base bleeding as two active flow control methods on aerodynamic drag reduction of Ahmed body with 35 degree rear slant angle. Suction in boundary layer is applied in order to delay flow separation by extracting flow particles with low kinetic energy near the model surface and the sucked air is blown into the wake of the model to increase the static pressure of the wake region. The location of suction is at the beginning of rear slant surface and the location of blowing is at the middle part of rear vertical part of the model. Moreover, the effect of change in control flow rate and suction and base bleeding area is investigated. Strong suction leads to drag reduction and when suction is accompanied by base bleeding more drag reduction can be achieved. Furthermore, for a constant control flow rate, smaller suction area and bigger base bleeding area yield to more reduction in drag.

Keywords   Aerodynamic drag reduction, Active flow control, Ahmed model.

چکیده   پسای آیرودینامیکی یکی از مهم‌ترین عوامل مصرف سوخت خودروها می‌باشد. ازطرفی، پسای فشاری به عنوان مولفه‌ی اصلی پسای کل در خودروها، ناشی از جدایش لایه‌ی مرزی از سطح خودرو است. در این پژوهش، اثر مکش و دمش به عنوان دو روش فعال کنترل جریان بر کاهش پسای آیرودینامیکی مدل احمد با زاویه‌ی شیب انتهایی ۳۵ درجه به صورت تجربی بررسی شده است. مکش در لایه‌ی مرزی با خارج کردن ذرات جریان با انرژی جنبشی کم جدایش جریان را به تعویق می‌اندازد. همچنین هوای مکیده شده به داخل ناحیه‌ی دنباله دمیده شده است تا فشار استاتیکی در ناحیه‌ی دنباله افزایش یابد. محل مکش ابتدای سطح شیب‌دار عقب مدل و محل دمش در قسمت عمودی انتهای مدل می‌باشد. علاوه بر این، اثر تغییر در دبی جریان کنترلی و سطح مقطع مکش و دمش در این مقاله مورد مطالعه قرار گرفته است. نتایج نشان می‌دهند اعمال مکش قوی باعث کاهش پسا می‌گردد و در صورتی که مکش و دمش همزمان صورت گیرد، کاهش بیشتری در نیروی پسا مشاهده می‌گردد. همچنین به ازای یک دبی ثابت برای جریان کنترلی هرچه سطح مکش کمتر و سطح دمش بیشتر باشد کاهش پسای بیشتری رخ می‌دهد.

1. INTRODUCTION

Because of high price of energy and the pollution due to fossil fuel consumption, reducing fuel consumption in vehicles has attracted a lot of interests. Among the factors which affect fuel consumption, the aerodynamic drag is one of the most important factors since more than half of the vehicle's energy is consumed in overcoming aerodynamic drag. Thus, the drag reduction is one

of the most interesting approaches in this matter. Aerodynamic drag consists of two main components: skin friction drag and pressure drag. Pressure drag accounts for more than 80 percent of the total drag. The pressure drag which highly depends on vehicle geometry is because of the boundary layer separation from rear window surface and formation of wake region behind the vehicle. The location of separation determines the size of wake region and consequently, it determines the value of aerodynamic drag.

Passive flow control strategies such as vortex generators are accompanied by vehicle geometry modifications which may not be favorable due to stylistic reasons. Applications of some passive control methods are already reported. Koike et al.[1] studied the effect of different configurations of vortex generators on drag reduction of a commercial car model. Porous media were also utilized to decrease the drag of a car model by Bruneau et al. [2]. Cooper [3] could reach drag reduction of up to 18 percent by implementation of base flaps in heavy vehicles. Researchers in NASA tested some panels in front of cars to affect the flow around the car and could decrease the aerodynamic drag, experimentally. However, their method may not be applicable in road cars [4].

On the other hand, active control methods such as suction and blowing are replacing passive methods not only in the field of vehicle aerodynamics, but also in other fields of aerodynamics. However, active methods still seem to be too academic and they may not be practical and more research should be done to optimize the control performances. Rotating cylinders were applied on a heavy vehicle tractor by Singh et al. [5]. They showed that utilizing rotating cylinders decreases wake region and consequently aerodynamic drag. Salari et al. [6] reported the effect of base bleeding on heavy vehicles. Nisugi et al. [7] did a numerical research on feedback flow control of a simplified car model. Furthermore, drag reduction by spanwise wall oscillation was studied and reported by Zhou and Ball [8]. In the recent years, new methods of active flow control have been investigated due to development of new technologies. For example, Brunn [9] et al. used MEMS actuators for active flow control of a car model. Flow control via plasma actuators is also another novel flow control method which is mainly applied in supersonic flows [10].

Since a real-life car shape is too complex for numerical simulations or experimental studies, simple car models are employed. One of these models is called Ahmed model after being employed by Ahmed et al. [11]. Ahmed model can generate fully three-dimensional regions of separated flow similar to what happens in real vehicle shapes. Precise experimental data for aerodynamic characteristics of Ahmed body are given by Leinhart and Becker [12]. This model has been interesting for CFD researchers and simulating flow over Ahmed body has been the subject of many papers in recent years [13-15]. In addition, Abbaspour and Jahanmiri investigated the effect of flow suction and base bleeding on drag reduction of Ahmed model, numerically [16, 17].

In most studies, two different slant angles 25° and 35° are considered where the critical angle of 30° lies between these two values. 30° is the angle at which the drag is maximum. The bottom surface of the Ahmed body is located at 50 mm above the ground. Figure 1 shows the geometry of Ahmed model.

In this paper, Ahmed model is used to study the effect of suction and base bleeding on drag reduction. In addition, different areas for suction and base bleeding and different control flow rates are studied.
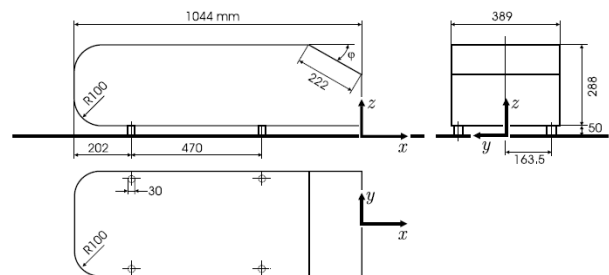

Figure 1. Geometry of Ahmed model

2. EXPERIMENTAL SETUP

Wind Tunnel: The experiments were carried out in a subsonic closed circuit wind tunnel with a closed test section. The test section had the cross section of 80 cm × 80 cm and the length of 200cm.

6-Component Balance: a 6-component balance capable of measuring lift, drag and side force and also pitching, rolling and yawing moments was utilized in this experiment. However, the only important parameter in this experiment which was measured by the balance, was the drag force on the model.

Suction Device: A suction device was utilized outside of the wind tunnel which sucked the air via a flexible tube. The suction flow rate was adjusted by a tap and a flow meter.

Blowing device: An air compressor was used out of the wind tunnel to blow the air into the wake region via the flexible tube. The blowing flow rate was also adjusted so that the suction and blowing flow rates were equal.

Model: As mentioned before, Ahmed model was used to study the effect of suction and base bleeding on drag reduction. Since the test section of the wind tunnel was small compared with the original size of Ahmed body, and to decrease the negative effect of solid blockage on the drag force measurement, a half scale Ahmed body was used. Instead, in order to satisfy the condition of similarity, the free stream velocity at the beginning of test section was adjusted twice the value of free stream velocity in the experiments on the full scale Ahmed model so that the Reynolds number based on model length became 2.8×10^6.

Since the outer dimensions of the model is very important, the model was designed in such a way that suction and base bleeding was possible without any change in the outer geometry of the model. Furthermore, it was possible to investigate the effect of the change in the suction and base bleeding area. The model is shown in Figures 2 and 3.
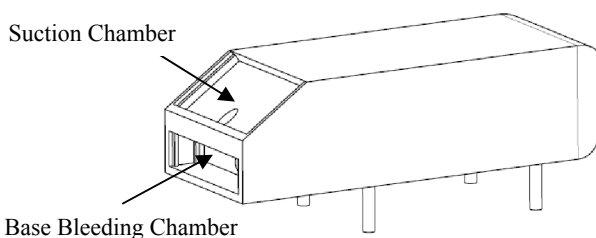

Figure 2. A scheme of model including suction and base bleeding chambers
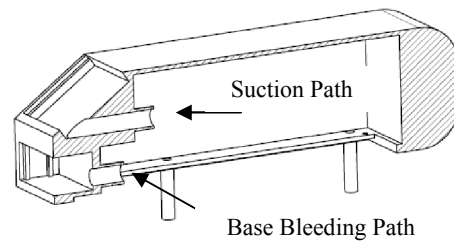

Figure 3. A cut view of model

As it can be observed, there are chambers in the rear slant part and rear vertical part which were covered by putting the porous surfaces -as suction and base bleeding surfaces- on their edges. Two different suction surfaces and two different base bleeding surfaces were studied in this paper which had different number of holes. The suction surfaces with two and four rows of holes are called S2 and S3, respectively. In addition, the base bleeding surfaces with two and four rows of holes are called B2 and B3, respectively. These surfaces are shown in the figures 4 and 5. Moreover, S1 and B1 are surfaces with no holes which were put on the suction and base bleeding chambers when it was intended to block suction or base bleeding completely. It is noteworthy that the stilts of the model were made hollow in order to let the sucked and blown air exit and enter the model, respectively.
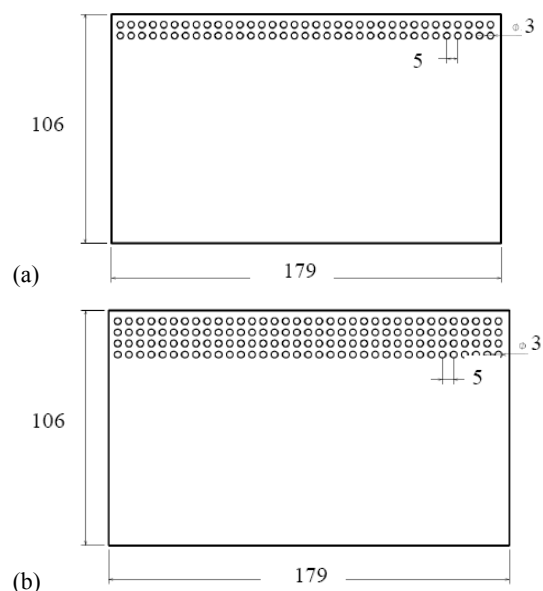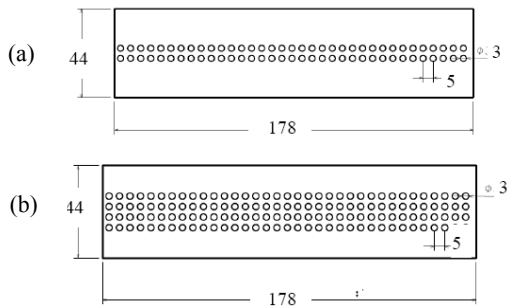

Figure 4. Suctio surfaces (a) S2, (b) S3

Figure 5 . Base bleeding surfaces (a) B2, (b) B3

The air was sucked at the holes of suction surface and then it passed through a flexible tube which in turn, left test section through one of the hollow stilts of the model toward the suction device. Also, the air blown by the air compressor entered the model via a flexible tube which passes through another hollow stilt of model.

3. EXPERIMENT RESULTS

At the first step, the drag coefficient of Ahmed model was obtained without applying flow control. The nonporous S1 and B1 surfaces were used to cover suction and base bleeding cambers thoroughly. The drag coefficient of Ahmed model in case with no control was obtained 0.312. This value is more than 0.260 which was reported by Ahmed et al.[3] (1984) who did their experiments in an open test section wind tunnel. The reason for this difference can be the high solid blockage of the model in the test section of wind tunnel in which our tests were carried out. To show the values of uncertainties, Figures 6 and 7 contain the error bars including standard deviation. The values of the uncertainties for figures 8 to 16 are of the same order with uncertainties in Figures 6 and 7. Here, the error bars for figures 8 to 16 are not drawn to avoid the complexity in figures contents. According to the error analysis, the maximum uncertainty in our experiments is less than 3 percent which is satisfactory enough.

After calculating the drag coefficient in case without flow control, different configurations of suction and base bleeding surfaces were installed on the model and the effect of flow control was investigated. Each configuration was named based on the name of suction and blowing surfaces. For instance, S2-B3 refers to the configuration in which the suction surface S2 and blowing surface B3 were installed on the model.

3.1. Effect of Boundary Layer Suction
Figures 6 and 7 show the effect of boundary layer suction on drag coefficient of Ahmed model when S2 and S3 surfaces were installed on the suction chamber, respectively. Here, no base bleeding took place. Both figures show similar trend of variation in drag coefficient. It can be observed that when the suction flow rate is less than 0.0021 m^3/s the drag coefficient increases and then by increasing the suction flow rate from 0.0021 m^3/s to 0.0056 m^3/s the drag coefficient decreases. It seems that when the suction is not strong enough to affect and deflect the free stream, applying suction can strengthen the secondary flow on the rear slant surface of the model and it increases the wake size and thereby, the drag is increased.
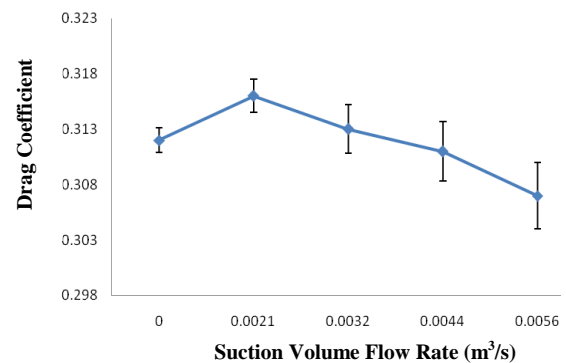

Figure 6. Variation of drag coefficient with Suction flow rate for the case S2-B1
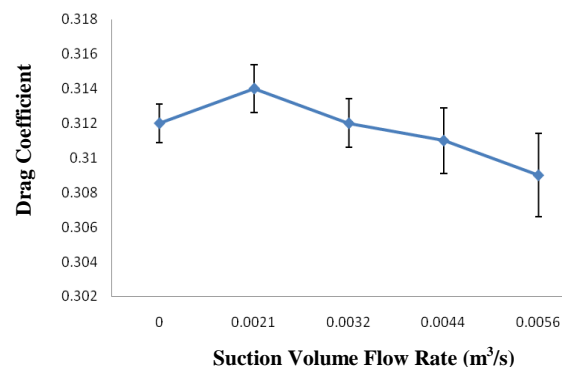

Figure 7. Variation of drag coefficient with Suction flow rate for the case S3-B1

3.2. Effect of Simultaneous Boundary Layer Suction and Base Bleeding In the next step, effect of simultaneous suction from the rear slant and base bleeding at the rear vertical part of the model was studied. Figures 8 and 9 show the comparison between the case in which only suction from surface S2 was applied and the cases in which in addition to suction from S2 and base bleeding at surfaces B2 and B3 were carried out, respectively.
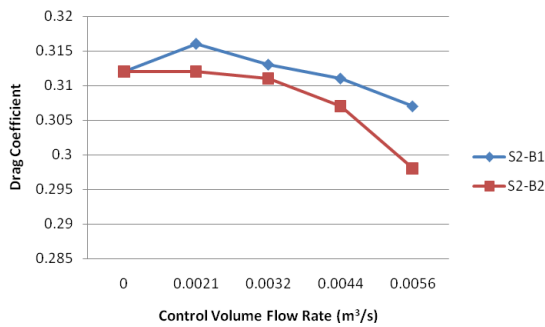

Figure 8. Variation of drag coefficient with control flow rate for the cases S2-B1 and S2-B2 (same suction surfaces)
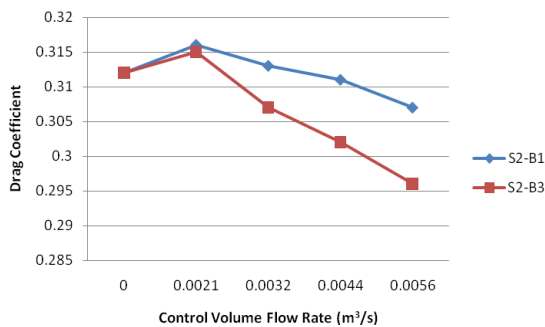

Figure 9. Variation of drag coefficient with control flow rate for the cases S2-B1 and S2-B3 (same suction surfaces)

Also, Figures 10 and 11 illustrate similar comparison for suction surface S3 and base bleeding surfaces B2 and B3. As it can be observed, when base bleeding and suction were simultaneously, more reduction in drag coefficient was achieved. That is because the base bleeding enhances static pressure in the wake region and therefore, the pressure difference between the front and rear of the model reduces.
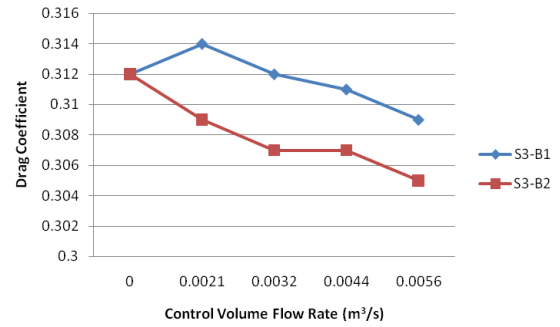

Figure 10. Variation of drag coefficient with control flow rate for the cases S3-B1 and S3-B2 (same suction surfaces)
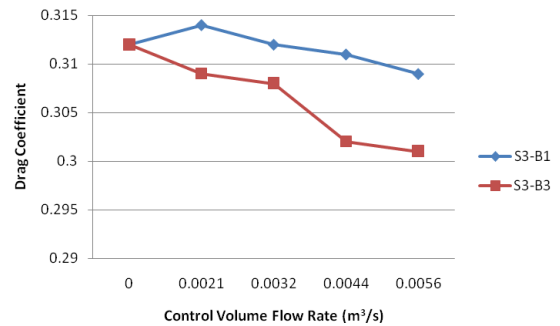

Figure 11. Variation of drag coefficient with control flow rate for the cases S3-B1 and S3-B3 (same suction surfaces)

3.3. Effect of Change in the Suction Area To study the effect of change in the suction area, Figures 12, 13 and 14 are presented. In each figure, the base bleeding area is fixed and the effect of suction areas S2 and S3 are compared with each other. The results show that when suction is strong enough (more than 0.0044 m^3/s), the less suction area leads to the less drag coefficient value.
for the cases S3-B1 and S3-B2 (same suction surfaces)

3.4. Effect of Change in the Base Bleeding Area The results presented in Figures 15 and 16 demonstrate that for control flow rates more than 0.0032 m^3/s, when base bleeding is carried out at bigger area(B3), more drag reduction is achieved than the case in which base bleeding is applied at smaller area(B2). This is because for a constant control flow rate, the static pressure of the flow

with bigger cross section is more than the static pressure of the flow with smaller cross section since the total pressure of both flows provided by the air compressor is constant.
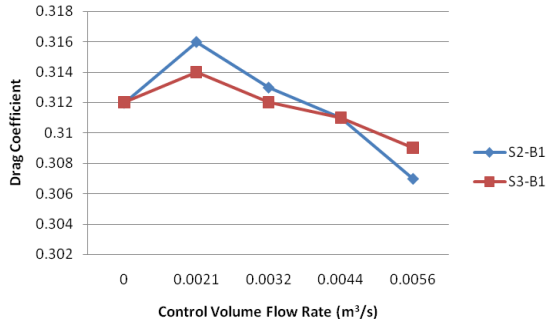

Figure 12. Variation of drag coefficient with control flow rate for the cases S2-B1 and S3-B1 (same base bleeding surfaces)
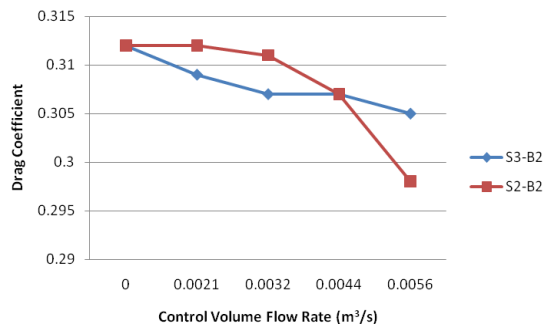

Figure 13. Variation of drag coefficient with control flow rate for the cases S2-B2 and S3-B2 (same base bleeding surfaces)
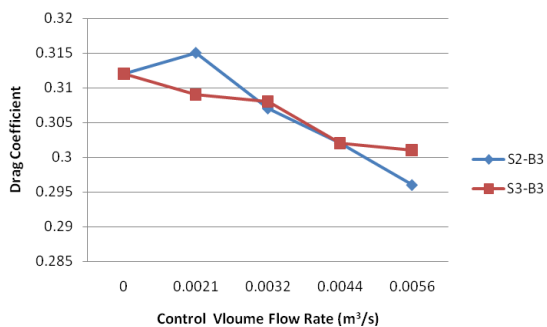

Figure 14. Variation of drag coefficient with control flow rate for the cases S2-B3 and S3-B3 (same base bleeding surfaces)
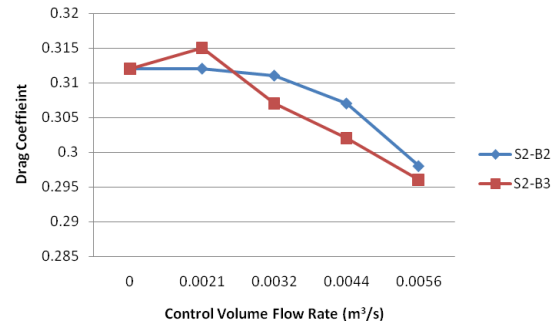

Figure 15. Variation of drag coefficient with control flow rate for the cases S2-B2 and S2-B3 (same suction surfaces)

4. NUMERICAL ANALYSIS

The numerical analysis of the flow has been done by the authors in their previous works [16-17]. Since the numerical simulations have been carried out in two dimensions, their results are not suitable for quantitative comparison but for qualitative comparison. Interestingly, the numerical results are verified by the experiments. In section 3.1 of this paper it is mentioned that for the low suction rates (less than 0.0021 m^3/s) an increase in drag coefficient may be observed. The numerical simulation shows that low suction rates strengthen the secondary flow on the rear slant and consequently the wake region is developed. Figure 17 shows the velocity vectors on the rear slant of Ahmed model with different suction flow rates. The simulations also show that for strong enough control flow rate, increasing control flow rate leads to reduction in wake size and drag force. This trend is illustrated via velocity contours around the model in Figure 18.
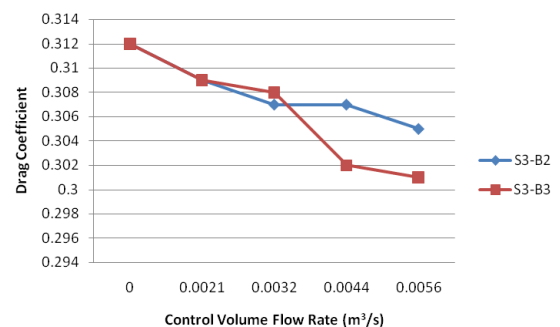

Figure 16. Variation of drag coefficient with control flow rate for the cases S3-B2 and S3-B3 (same suction surfaces)

Figure 17. Velocity vectors on the rear slant of the model for different magnitude of suction rate: a) without suction, b) weak suction and c) strong suction

Figure 18. Wake region reduction by increasing control flow rate

5. CONCLUSION

In this paper, the effect of suction and base bleeding which are known as two different active flow control methods on drag reduction of Ahmed model was investigated experimentally and the experimental results confirm the previous numerical simulation works. Some changes were devised in the model to make suction and base bleeding possible without changing the outer geometry of the model. Suction was applied at the beginning of rear slant surface and base bleeding was applied at the middle of rear vertical part of the model. Furthermore, it was feasible to evaluate the effect of different areas for suction and base bleeding. Finally, the following conclusions can be drawn from the obtained results:

1. Boundary layer suction at the beginning of rear slant surface of Ahmed model can reduce the drag. However, if the suction is less than 0.0021 m^3/s, it may lead to the increment of drag. This is due to the fact that weak suction intensifies the secondary flow on the rear slant surface of the model rather than affecting free stream.

2. Simultaneous suction and base bleeding yields to more reduction in drag than the case in which only suction is applied. This conclusion verifies the data from numerical simulations previously done in Reference 9.

3. In case of strong suction (more than 0.0044 m^3/s), similar to the data from Reference 8, the less the area of the suction, the less value of the drag is achieved.

4. Base bleeding at the bigger area leads to more static pressure recovery in the wake region and thereby it decreases the aerodynamic drag. This also verifies the results obtained in Reference 9.

5. As it is obvious from the above results, the maximum drag reduction was achieved when the smaller suction area(S2) and the bigger base bleeding area(B3) were applied with the maximum possible control flow rate (0.0056 m^3/s). In this case, 4 percent reduction in drag coefficient was observed.

6. It should be reminded that this reduction in drag could be increased if the experimental set-up is well selected and any leakage of flow is prevented.
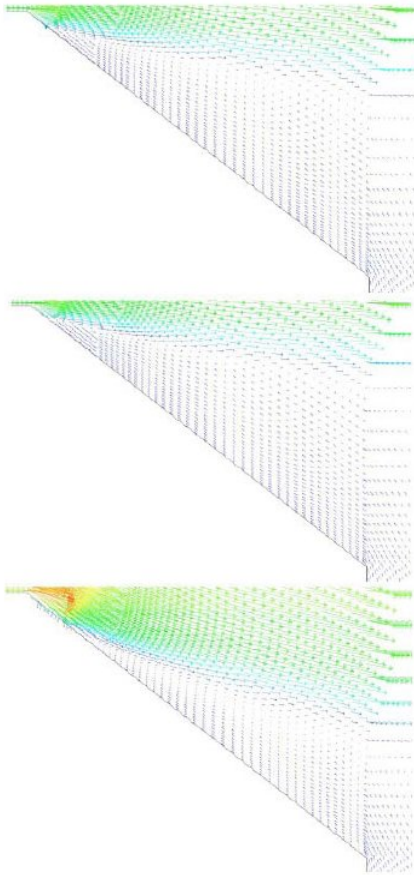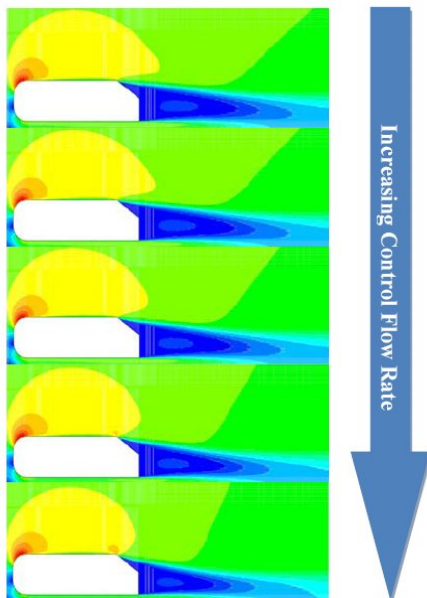
6. REFERENCES

1. Koike, M., Nagayoshi, T. and Hamamoto, N., "Research on aerodynamic drag reduction by vortex generators", *Mitsubishi motors technical review,* No.16, (2004), pp. 11-16.
2. Bruneau, C. H., Gilliéron, P. and Mortazavi, I., "Flow manipulation around the Ahmed body with a rear window using passive strategies", *Comptes Rendus Mecanique,* Vol. 335, No. 4, (2007), pp. 213-218.
3. Cooper, K., "The effect of front–edge rounding and rear-edge shaping on the aerodynamic drag of bluff vehicles in ground proximity". *SAE paper*, 850 288,(1985).
4. Pamadi, N. B., Taylor, W. L. and Leary, O. T, "A method for the reduction of aerodynamic drag of road vehicles" NASA technical memorandum 102589, 1990.
5. Singh, N. S., Rai, L., Puri, P. and Bhatnagar, A., "Effect of moving surfaces on the aerodynamic drag of road vehicles". *Proceeding IMechE. Part D: Journal of Automobile Engineering,* Vol. 219, (2005).
6. Salari, K., Ortega, J. and McCallen, R. "Heavy Vehicle Drag Reduction Devices: Computational Evaluation & Design". *DOE Heavy Vehicle Systems Review*, (2006), pp. 18-20,.
7. Nisugi, K., Hayase, T. and Shirai, A., "Fundamental study of aerodynamic drag for vehicle with feedback flow control", *JSME International Journal, series B,* Vol. 47, No. 3, (2004), pp. 584-592.
8. Zhoe, D. and Ball, K. S. "Drag reduction by spanwise wall oscillation". *International Journal of Engineering*, Vol. 21, No.1, (2008), pp. 85-104
9. Brunn, A., Wassen, E., Sperber, D., Nitsche, W. and Thiele, F. "Active drag control for a generic car model", R. King (Ed.), Vol.95, Active Flow Control, NNFM, (2007), 247-259.
10. Shang, J. S., Surzhikov, S.T., Kimmel, R., Gaitonde, D., Menart, J. and Hayes, J., "Mechanisms of plasma actuators for hypersonic flow control" *Progress in Aerospace Sciences,* Vol. 41, (2005), pp. 642–668.
11. Ahmed, S.R. and Ramm, G., "Some Salient Features of the Time-Averaged Ground Vehicle Wake", *SAE Technical Paper,* 840300, 1984.
12. Lienhart, H., Stoots, C. and Becker, S., "Flow and turbulence structures in the wake of a simplified car model", *SAE technical paper,* (2003).
13. Guilmineau, E., "Computational study of flow around a simplified car body", *Journal of wind engineering and industrial aerodynamics*, Vol. 96, (2008), pp. 1207-1217.
14. Fares, E., "Unsteady flow simulation of the Ahmed reference body using a lattice Boltzmann approach", *Journal of Computers & Fluids,* Vol. 35, (2006), pp. 940-950.
15. Hinterberger, C., Garcia-Villalba, C. and Rodi, W., "Large eddy simulation of flow around Ahmed body", The Aerodynamics of heavy vehicles: Trucks, Buses and Trains, Springer, 2004.
16. Abbaspour, M. and Jahanmiri, M., "Two dimensional investigation of the effect of boundary layer suction on drag reduction of a car model", *9th annual Iranian aerospace society conference,* The Science and Research Branch-Islamic Azad University, Tehran, Iran, 2010 (printed in persian).
17. Abbaspour, M. and Jahanmiri, M., "Computational Study of Aerodynamic Drag Reduction on AhmedBody Using Active Strategies", *13th Annual & 2nd International Fluid Dynamics conference*, Shiraz University, Shiraz, Iran, 2010