International Journal of Mechanical, Industrial and Aerospace Sciences
ISSN: 2517-9950
Vol:9, No:1, 2015

Vehicle Aerodynamics: Drag Reduction by Surface Dimples

C. K. Chear, S. S. Dol

***Abstract*—**For a bluff body, dimples behave like roughness elements in stimulating a turbulent boundary layer, leading to delayed flow separation, a smaller wake and lower form drag. This is very different in principle from the application of dimples to streamlined body, where any reduction in drag would be predominantly due to a reduction in skin friction. In the present work, a car model with different dimple geometry is simulated using k-ε turbulence modeling to determine its effect to the aerodynamics performance. Overall, the results show that the application of dimples manages to reduce the drag coefficient of the car model.

***Keywords*—**Aerodynamics, Boundary Layer, Dimple, Drag, Kinetic Energy, Turbulence.

I. Introduction

THE successful application of dimples on a golf ball has brought inspiration to the vehicle aerodynamics engineers. A golf ball with dimple surface can travel higher and further than a smooth surface golf ball when subjected to identical force (i.e. club swing). Dimples on the golf ball induce turbulence at lower Reynolds number (a non-dimensionless number that determines whether the flow is laminar or turbulent) providing extra momentum or energy to the boundary layer causing delay in flow separation, as shown in Fig. 1 [4], [6]. This phenomenon causes smaller wakes area (i.e. swirling flow region) behind the ball thus reducing the total drag [2].

Every single body that moves in fluid will experience drag. Basically, the drag force can be divided into two different types which are pressure drag and friction drag [4], [6]. Friction drag comes from friction between the fluid and the contacted surface. Friction drag is related with the development of boundary layer (thin layer of flows close to the surface due to fluid viscosity) and the Reynolds number. Pressure drag forms due to the eddying motions that are set up in the fluid by the passage of the body. Pressure drag is related with the formation of a wake (i.e. shape or form of the body). When drag force of a body is dominated by friction drag, the body will be defined as a streamline body (i.e. fish, bird, or an airfoil that is in small or zero angles of attack). On the other hand, the drag force of a bluff body (i.e. brick, golf ball or an airfoil that is in large angle of attack) is dominated by pressure drag.

C.K. Chear is with the Department of Mechanical Engineering, Curtin University, Miri, Sarawak, Malaysia.
S. S. Dol is with the Department of Petroleum Engineering, Curtin University, Miri, Sarawak, Malaysia (e-mail: sharulsham@curtin.edu.my).

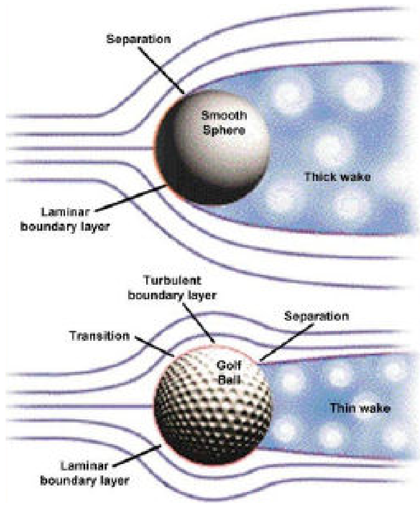


Fig. 1 Dimple pattern on golf ball

In a famous science entertainment television program Mythbusters in 2009 has made an experiment on applied dimples on the car surface (Fig. 2). In the program, Jamie and Adam have created dimples on a car surface with clay. They found out that there was increased of fuel consumption efficiency. The efficiency was increased by 11% when compared to a normal surface car, which was considered as a major improvement. Although it was not an official experiment or research, but it has shown that there is a possibility to apply dimples on the car surface or even other transportation mode like trains and aircrafts to reduce fuel consumption.

Fig. 2 The Mythbusters crew experimented with dimples on this Ford Taurus

Several previous studies (experimental and numerical) observed that dimples on an airfoil create extra turbulence to delay the boundary layer separation. The effect was improved when the airfoil was at angle of attack. For example, [5] has suggested a smart dimple matrix over the airfoil which will sense boundary layer separation and arrange dimple in the least drag and high lift configuration, as shown in Fig. 3. German researchers proved an effect which could revolutionize the design of ships, vehicles and aircrafts. Experiments in the wind tunnel showed: dimples in the surface of a body massively reduce the flow resistance. They came to an unexpected result: the train with dimples has 16% less frictional resistance, depending on the speed. The finding seems contradicting the theorem that says surface roughness (dimple) increases the surface area thus friction. Moreover, the drag reduction was overcome by the newly appearing pressure force contribution due to surface unevenness [3].

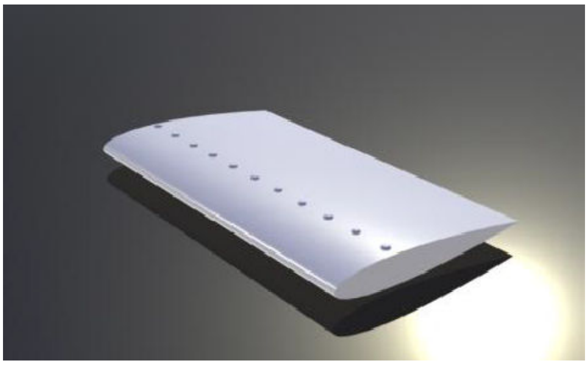


Fig. 3 Dimples matrix as suggested by [5]

So, will the dimples bring a significant effect to streamline body (i.e. vehicle design) as it brings to the bluff body? Adding a dimple on a streamlines body might help to delay the flow separation and reduce the size of the wake but it might be increase the friction drag as a trade-off. As the subject is enormously important to save energy and to protect the global environment, studies to further investigate the effect of dimple on the aerodynamics of the car body have been made with the aim to determine how much changes the dimple will bring to the coefficient of drag.

II. Methodology

A. Car Model

In this work, Ahmed body car model with 25 degree slant angle will be used. Ahmed body car model is a simplify car model for accurate flow simulation but still retain with the significant features of a car which are curved forebody, straight center section and the angled rear end. It is a typical bluff body which commonly using for the simulation to study the flow past a car [1]. With the dimension details from Ahmed research, the model will be drafted by using Autodesk AutoCAD software. After the default model is drafted, it will be added with different ratio of dimple at the near back of the roof. The body geometry is shown in Fig. 4.

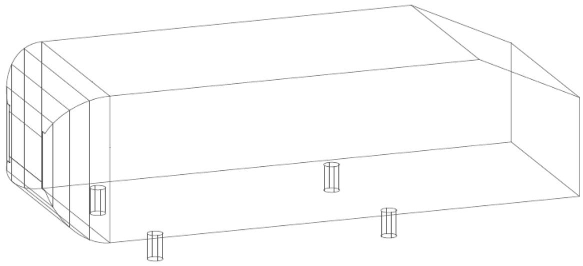


Fig. 4 Isometric view of the Ahmed body car model

B. Dimple Ratio

In order to have more precise result, a parameter called dimple ratio (DR) has been introduced. Basically, dimple ratio is the ratio between the depths of the half dimple over the print diameter of the dimple (Fig. 5). In this work, car model with be simulated with the DR of 0.05, 0.2, 0.3 0.4 and 0.5.

$$\text{Dimple ratio, DR} = \text{depth}, h / \text{diameter}, D \qquad (1)$$

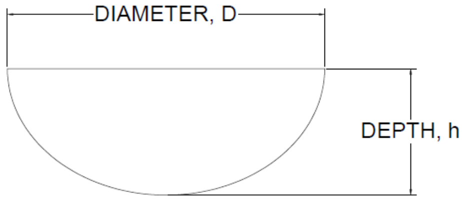


Fig. 5 Dimple ratio

C. Meshing

The meshing is done by using ANSYS meshing software. Tetrahedral meshing is used for the analysis (Fig. 6). Table I shows the setting that has been applied to all models. It shows that the skewness and the orthogonal quality of the meshing are within the acceptable region of simulation accuracy.

TABLE I
QUALITY OF MESH FOR ALL MODELS

Model	Nodes	Elements	Skewness	Orthogonal Quality
Smooth surface	142564	782743	0.222	0.862
DR = 0.05	319366	1798765	0.221	0.863
DR = 0.2	282862	1578882	0.221	0.863
DR = 0.3	283411	1580105	0.221	0.863
DR = 0.4	284911	1586428	0.221	0.863
DR = 0.5	280090	1560387	0.221	0.863

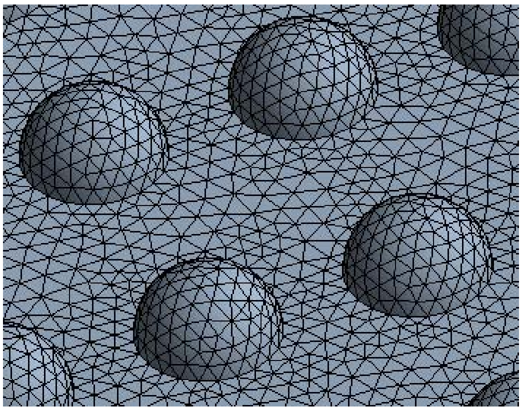


Fig. 6 Tetrahedral mesh around dimples

D. Turbulence Model

In this paper, k-ε model in ANSYS Fluent has been chosen for turbulence modeling. k-ε model consists of two transport equations to represent the turbulent properties of the flow. The first transport variable is the kinetic energy term, k. It is the variable to determine the turbulent energy production.

$$\partial/\partial t\,(\rho k) + \partial/\partial x_i\,(\rho k u_j) = \partial/\partial x_j\,[(\mu + \mu_t/\sigma_k).\partial/\partial x_j] + G_k + G_b - \rho\varepsilon - Y_M + S_k \quad (2)$$

The second transport variable is the turbulent dissipation term, ε. It is the variable to determine the scales of the turbulence.

$$\partial/\partial t\,(\rho\varepsilon) + \partial/\partial x_i\,(\rho\varepsilon u_j) = \partial/\partial x_j\,[(\mu + \mu_t/\sigma_\varepsilon)\,\partial_\varepsilon/\partial x_j] + \rho C_1 S_\varepsilon - \rho C_2[\varepsilon^2/(k + \sqrt{v\varepsilon})] + C_{1\varepsilon}.\varepsilon/k\,.C_{3\varepsilon}.G_b + S_\varepsilon \quad (3)$$

In the equations above, G_k is the generation of turbulence kinetic energy due to the mean velocity gradients. G_b is the generation of turbulence kinetic energy due to buoyancy. Y_M is the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate. $C_{1\varepsilon}$andC_2,and $\sigma_k$andσ_ε are constants and turbulent Prandtl numbers for k and ε respectively. $S_k$andS_ε are user defined source terms.

E. Boundary Conditions

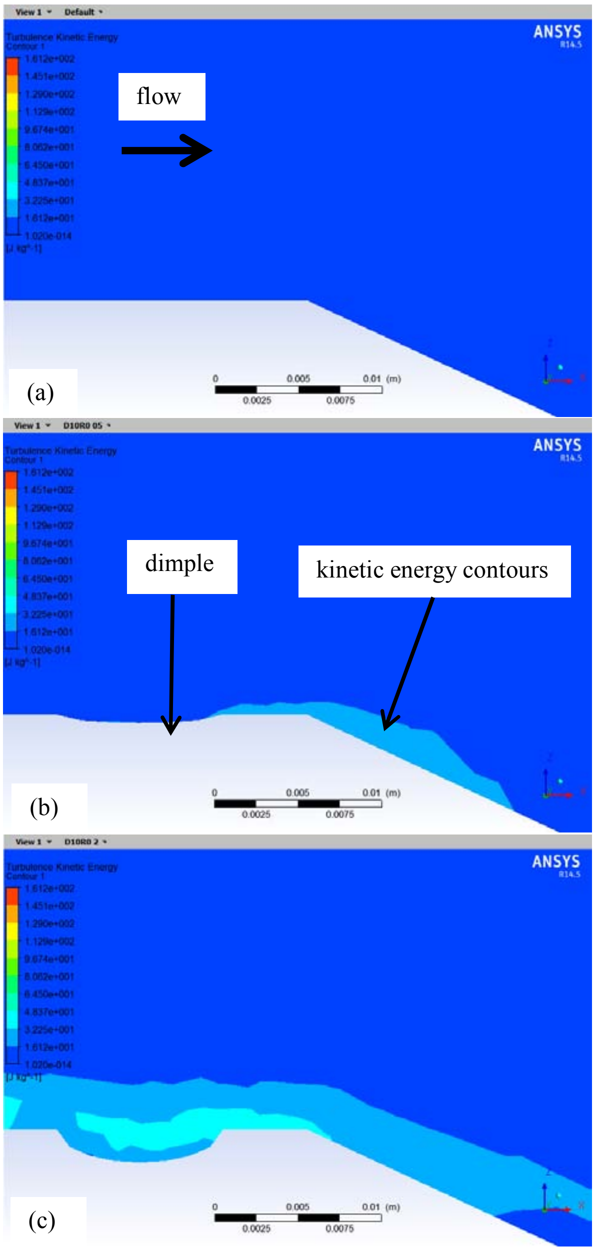
In this work, the air velocity has been set to 40 m/s at the inlet that gives the Reynolds number as;

$$Re = \frac{\rho UA}{\mu} = \frac{1.2\times 40\times 0.065741}{1.789\times 10^{-5}} = 176387 \quad (4)$$

where ρ is the density of air; is the flow velocity; A is the frontal area of the car model; μ is the dynamic viscosity of air.

III. Results

For the model without application of dimple, there is insignificant turbulent kinetic energy on the car surface (Fig. 7 (a)). When compared with the model without dimple, turbulent kinetic energy is generated within the dimple and at the vicinity of the dimple edge (Figs. 7 (b)-(f)). These results suggest that the boundary layer flows manage to go further before the flow separation takes place. The geometry of dimple changes the kinematics and dynamics of flow. Complex interaction between the turbulent fluctuating flow and the mean flow escalates the turbulence quantities and the boundary layer gains momentum from this turbulent kinetic production. The maximum level of turbulent kinetic energy occurs at DR = 0.4 as shown in Fig. 7 (e).



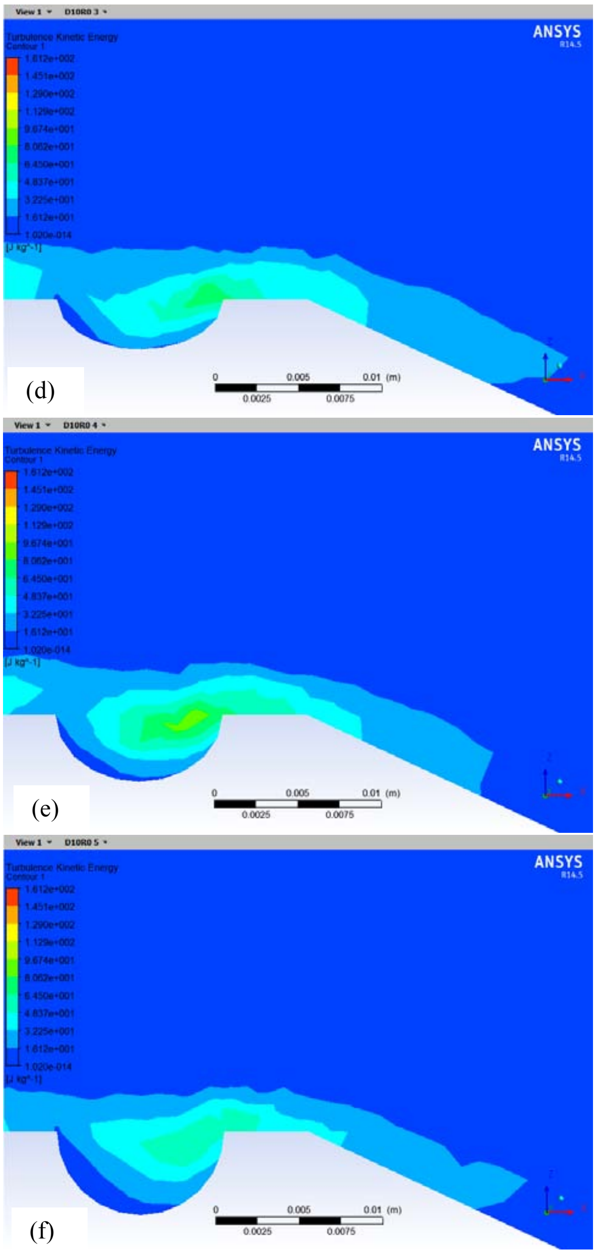


Fig. 7 Turbulence kinetic energy contours (a) DR = 0; (b) DR = 0.05; (c) DR = 0.2; (d) DR = 0.3; (e) DR = 0.4; (f) DR = 0.5

TABLE II
COMPARISON OF DRAG COEFFICIENT

Model	Drag Coefficient	Percentage Difference (%)
Smooth surface	0.261	-
DR = 0.05	0.257	1.32
DR = 0.2	0.259	0.55
DR = 0.3	0.260	0.07
DR = 0.4	0.255	1.95
DR = 0.5	0.258	0.84

IV. Conclusion

All the dimple ratio models give smaller coefficient of drag, C_D compared to the model without dimple (Table II). The coefficient of drag, C_D is reduced by 1.95% for the model with DR = 0.4. The results are encouraging since the simulation is only based on one dimple. Different parameters like dimples position, number of dimples, dimples orientation will be tested in order to fully understand the performance of the dimple application on vehicle aerodynamics.

References

[1] Ahmed, S.R., Ramm, G. & Faltin G. (1984). Some salient features of the times-averaged ground vehicle wake. SAE Society of Automotive Eng., Inc, 1(840300):1 – 31.

[2] Dol, S.S., Kopp, G.A. & Martinuzzi, R.J.M. (2008). The suppression of periodic vortex shedding from arotating circular cylinder. Journal of Wind Engineering and Industrial Aerodynamics 96, 1164 – 1184.

[3] Hermann, L., Michael, B. & Cagatay, K. (2008). Drag Reduction by dimples? A Complementary Experimental/Numerical Investigation. International Journal of Heat and Fluid Flow 29(3), 783 - 791.

[4] Munson, B.R, Young, D.F. & Okiishi, T.H. (2002). Fundamentals of Fluid Mechanics, 4th Edition, John Wiley and Sons.

[5] Srivastav, D. (2012) Flow control over airfoils using different shaped dimples. International Conference on Fluid Dynamics and Thermodynamics Technologies (p. 92-97). Singapore: IACSIT Press.

[6] White, F.M. (2003). Fluid Mechanics, 5th Edition McGraw Hill.