INTERNATIONAL JOURNAL OF ADVANCE RESEARCH, IDEAS AND INNOVATIONS IN TECHNOLOGY

ISSN: 2454-132X
Impact factor: 4.295
(Volume3, Issue5)
Available online at www.ijariit.com

CFD Based Quantification of Aerodynamic Performance of an Automotive with Roof Dimples

Akshay Ashok Kumar
Department of Mechanical Engineering, University of Leeds, UK
kumar.akshay11293@gmail.com

Abstract:* *The impact of aerodynamics on vehicle performance has gained greater importance in the modern days. The effects of drag and lift forces on automotive have been well accounted for and engineers are relentlessly working on minimising the negative implications of these aerodynamic forces. The dimples on golf balls have been proven to reduce drag and are given credit for a golf ball's long flight. The use of dimples to reduce drag has often inspired many aerodynamists and this research aims to quantify the drag and lift forces on an automotive with dimples on its roof. A wide range of simple ratios, arrangements, and velocities have been taken into consideration and successful reduction in drag and lift coefficients has been reported.

Keywords: CFD, Automotive Aerodynamics, Ahmed Body, Golf Ball, Drag, and Lift.

I. INTRODUCTION

In the early 1900s when the automotive industry was in its formative days, engine and chassis development technologies were given priority over the aerodynamic performance of a vehicle [1]. However, it was towards the latter half of the 20th century, those aerodynamics took the center stage as the demand for lighter, faster and more energy efficient vehicles increased. This shift in focus is reflected in the evolution of automotive shapes from huge and box-like looks in the early 1900s to extremely sleek and aesthetic appearance of the modern cars [1].

When an automotive is set into motion, it is acted upon by resistive forces caused due to its interaction with the atmosphere. Aerodynamics is the branch of science which deals with such interactions and the forces which are caused due to this interaction are termed as aerodynamic forces. The resultant horizontal force which resists the forward motion of the vehicle is called the drag force and the resultant vertical force which enables the wheels to maintain traction with the road is termed as the downforce [2].

The aerodynamic requirements of every automotive are discrete and are governed by its application. As a common practice, passenger car designers ensure that the drag force is maintained minimum as it has a direct implication on the fuel economy. This drag reduction might come at the expense of a reduced downforce, but the latter is of less significance in passenger cars since the vehicle speed is not high enough to lift the wheels off the ground. However, a completely reversed approach has to be adopted for the high-performance cars (race cars), where it is more important to ensure that the wheel maintains traction with the road when the vehicle speed is very high. The drag reduction is given less priority since the powerful engines of the performance cars can easily overcome the resistances and fuel economy is of least importance to a race car. Modern aerodynamic devices such as wings and spoilers have been developed to increase the downforce and are hence termed as negative lift devices [2]. Thus it has to be appreciated that a trade-off has to be achieved between drag and downforce and often improvement in one comes at the expense of the other parameter.

One of the most significant features of a golf ball is its dimples. The dimples have been proven to reduce drag, thus enabling the golf ball to have a longer flight when compared to a simple spherical ball [3]. This has inspired engineers to investigate the correlation between the aerodynamics of a golf ball and that of an automotive and this research is also one such attempt. The Computational Fluid Dynamics (CFD) involves computer-based simulations and analysis of systems involving fluid flow and heat transfer [4]. This computer-based approach has replaced conventional wind tunnel tests in automotive aerodynamics.

The aim of this research is to perform a CFD based quantification of the aerodynamic performance of an automotive with dimples on its roof i.e., to investigate whether producing dimples on the roof of an automotive will have any effect on drag or downforce acting on the vehicle.

II. LITERATURE REVIEW

An extensive review of the literature was carried out in order to understand the outcomes of some of the research works previously carried out in this domain. Extensive knowledge on the flow features around a car and the aerodynamics of a golf ball was found to be important for the research.

One of the most significant features of a golf ball is its dimples. The basic aerodynamic behaviour of a sphere is much different from other bluff bodies such as rectangular flat plates or even the surface of a car [3]. The drag coefficient of a spherical ball largely depends on the Reynolds number of the flow. At lower Reynolds numbers, the coefficient of drag is very high (subcritical flow regime) and then as the Reynolds number increases the coefficient of drag undergoes a drastic reduction (critical regime) as shown in Figure 1 [3]. This steep reduction in drag coefficient is attributed to the transition of boundary layer from laminar to turbulent states, thereby energising the boundary layers to overcome the adverse pressure gradient at the back of the ball [6].

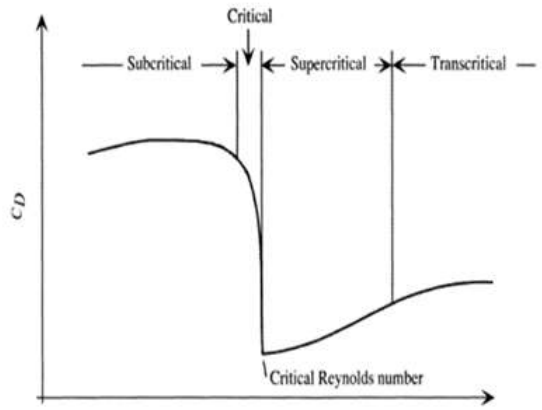

Figure 1 Flow Regimes over a Sphere [6]

The dimples on the golf ball tend to reduce the critical Reynolds number thereby causing an early transition from laminar to turbulent region. As a result, the separation of flow occurs further downstream for a dimpled ball causing a lower drag coefficient [3]. When the golf ball has no spin imparted on it, this reduced drag caused by the dimples is the reason for its longer and faster flight [7]. Dimples on golf balls tend to create a positive Magnus Effect at higher spin rates [3]. Thus the dimples on the golf ball improve the aerodynamic behaviour in normal situations and while spinning.

Dimples have been widely used in heat transfer application such as the boiler tubes. Dimples tend to increase the rate of heat transfer by encouraging flow separation (and thereby mixing) as the flow exits the dimple depressions [9]. The dimples tend to increase the surface area and rate of convection thereby increasing the Nusselt number [10]. Most of the previously conducted research works identify the depth of the dimples, the area it occupies and its arrangements tend to influence the drag characteristics. A 20% reduction in drag was reported by *Alekseev et al (1998)* [11] when shallow dimples were tested on flat plates. It was suggested that cross stream vortices tend to decrease drag which was later disproved by the work of *(Olaf) van Campenhout et al (2016)* [12]. When a single dimple was tested on the vehicle geometry, *Chear and Dol (2015)* [13] reported a 1.95% drag reduction at higher dimple ratio. Experimental works of *Burgees.N.K et al (2005)* [14], *Chen.Y et al (2012)* [15] and *Ligrani.P.M et al (2005)* [16] showed that friction drag increases with dimple depth while *Won.S.Y et al (2005)* [17] reported cross stream vortices at higher dimple ratios. It was also reported that large-scale unsteadiness is created due to vortex structures within deeper dimples [16]. On the contrary, *Lienhart. H (2010)* [18] showed that there is no change in drag coefficient and that the dimples are of no significant relevance to flow over flat plates.

One of the most common conclusions of all the researchers was that shallow dimples are the most likely to reduce the drag. *Isaev.S.A (2010)* [19] and *Alekseev et al (1998)* [11] commented that the flow separation is directly proportional to the dimple depth. Interestingly, the depth of the dimples also influences the performance of golf ball and *Chowdhury.H et al (2016)* [20] showed a linear increase in drag with dimple depth.

III.METHODOLOGY

A. Geometry and Fluid Domain

The fluid flow is governed by the following set of partial differential equations, known as the Navier Stokes Equations, represented by the conservation of mass [20]

$$\frac{\partial \rho}{\partial x} + \nabla \mathbf{u} = 0 \qquad \text{(Equation 3.1)}$$

and the conservation of momentum in each of the three Cartesian coordinates

$$\frac{d(\rho u)}{dt} + \nabla(\rho u \boldsymbol{u}) = -\frac{\partial P}{\partial x} + \nabla(\mu \nabla u) + S_{mx} \quad \text{(Equation 3.2)}$$

$$\frac{d(\rho v)}{dt} + \nabla(\rho v \boldsymbol{u}) = -\frac{\partial P}{\partial y} + \nabla(\mu \nabla v) + S_{my} \quad \text{(Equation 3.3)}$$

$$\frac{d(\rho w)}{dt} + \nabla(\rho w \boldsymbol{u}) = -\frac{\partial P}{\partial z} + \nabla(\mu \nabla w) + S_{mz} \quad \text{(Equation 3.4)}$$

Where u,v,w are the velocities in x,y and z directions respectively, ρ is the density of the fluid, t is the time, p is the pressure, $\boldsymbol{u}$ is the velocity vector, μ is the dynamic viscosity and S_{mx}, S_{my}, S_{mz} are the momentum source terms in x, y and z directions

The geometry that would be used for the analysis in this research would be an Ahmed Body with a slant angle of 25^0, the dimensions of which have been shown in Figure 2. The external shape of the solution domain is defined in terms of the length of the vehicle which is 1044mm. An elliptical flow domain has been used in this research since this reportedly reduces the number of elements by 20% thus reducing the computational expense of the simulation.

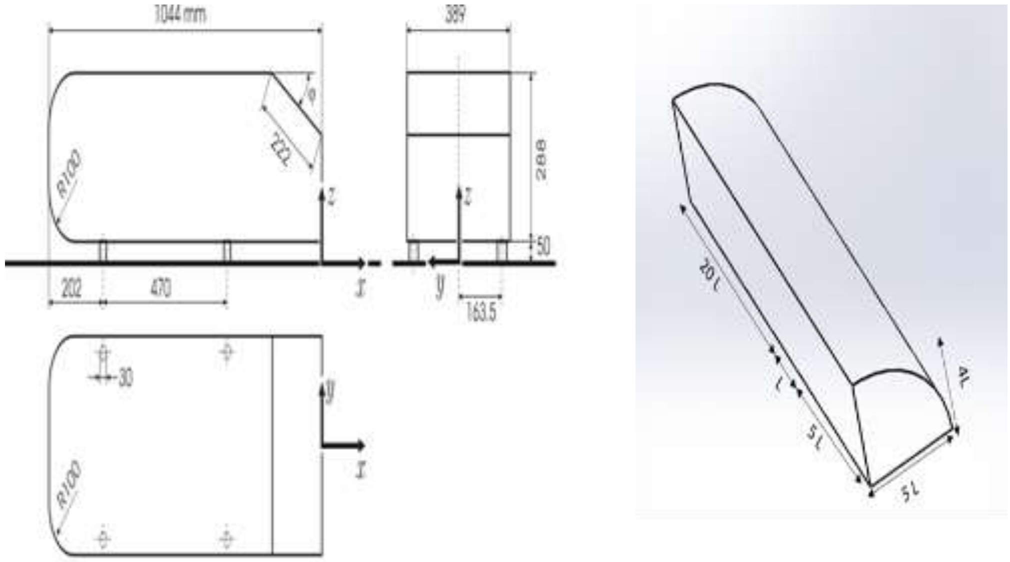

Figure 2. The Dimensions of the Ahmed Body [22] where φ is the slant angle and the elliptical flow domain [21] where L is the vehicle length

- **Dimple Ratio (DR)** is defined as the ratio of the depth of the dimple to its diameter, often expressed in percentage. Four different dimple ratios are considered in this research viz., 5%, 10%, 20% and 30%.
- **Coverage Ratio (CR)** is the ratio of the total surface area of the dimple to that of the surface area of the roof. Two different coverage ratios were taken into consideration viz., 40% (small but many dimples) and 70% (large and few dimples).

The following Figure 3 shows the arrangement of the dimples and the corresponding dimensions are presented in Table 1.

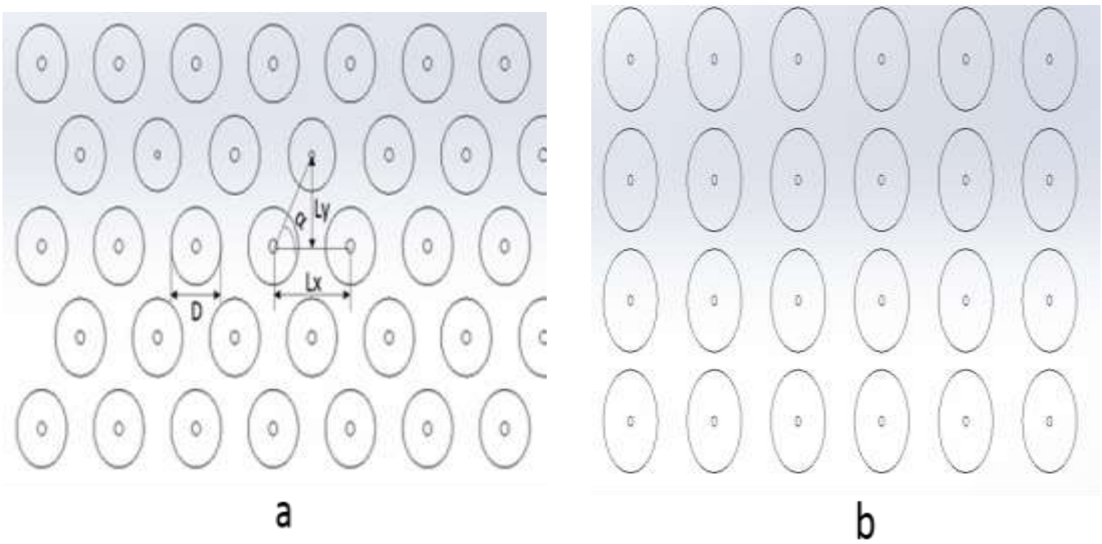

Figure 3. The Dimple Arrangements (a) Staggered (b) Parallel

TABLE I
DIMENSIONS AND DETAILS OF DIMPLE ARRANGEMENT

Parameter	Symbol	CR 40%	CR 70%
Dimple Diameter	D	16.26 mm	34 mm
Number of Dimples	-	126	48
Linear Spacing between each dimple	Lx	50mm	72.22 mm
Spacing between each row	Ly	38mm	72mm
Angular Spacing between rows (only for staggered)	α	56.66^0	63.37^0

B. *Meshing Strategy*

Asymmetry plane has been used which is a widely adopted practice in 3D simulations as it would bring down the number of elements by 30%. Unstructured tetrahedral meshes were used and the local mesh settings such as face and body sizing have been adjusted to achieve three levels of refinement. The regions away from the car body are left coarse with an element size of 38mm. A body of influence was created around the region close to the car body and the element size was brought down to 16mm. The element size on the faces of the car body was reduced to 8mm. A thin inflation layer was created around the car in order to capture the boundary layer separation with greater accuracy. The skewness of all the meshes was in the range of 0.75-0.8 which is good for tetrahedral meshes. The mesh refinement and inflation layers have been shown in figure 4 and 5.

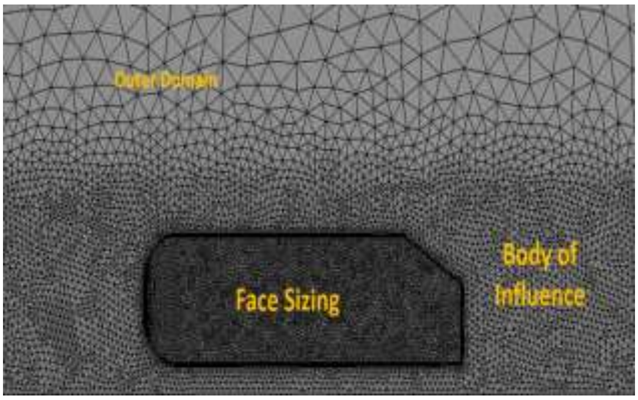

Figure 4. The various degrees of mesh refinement

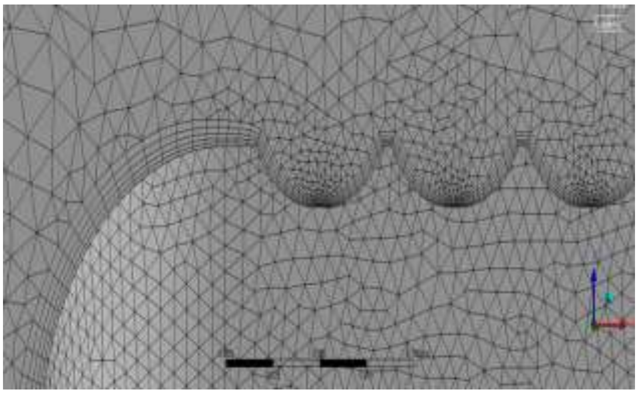

Figure 5. Inflation layer around a model with dimples

C. *Boundary Conditions*

The analysis was started at 40m/s as this is the standard start point velocity for any Ahmed Body analysis [22]. The analysis was further carried out at 60m/s and 80m/s. The turbulence model used is Realizable K-e with the wall function-based approach, since this model yields accurate results in cases involving flow separation and adverse pressure gradients [23]. The following table 2 is a brief account of the solver settings and the boundary conditions.

TABLE II
SOLVER SETTINGS AND BOUNDARY CONDITIONS

Setting	Option / Value	Justification
Solver Type	Pressure Based	Incompressible flow
Turbulence model	Realizable k-e	Ideal for cases with adverse pressure gradient and computationally affordable
Near Wall Treatment	Non-Equilibrium wall function	Better use of fluid laws at wall and will not treat wall as a flat plate
Inlet Velocity	40,60,80 m/s	Experiments on Ahmed body start from 40m/s
Inlet Turbulent Intensity	0.25%	External flow with no turbulence
Inlet Turbulent Length Scale	0.07308m	7% vehicle length [20]
Outlet Turbulent Intensity	0.25%	Same as Inlet
Outlet Turbulent Length Scale	0.07308m	7% vehicle length [20]
Reference Area	0.056016 m^2	Ahmed Body height frontal area
Coupling Scheme	SIMPLE	Incompressible flow and iterative technique is required
Gradient	Green-Gauss Node	Accurate discretization for unstructured tetrahedral meshes
Upwind Scheme	Second Order	High Accuracy of iteration results
Absolute Criteria	1e-10	For better accuracy of results
Number of Iterations	10000	Convergence is expected to occur well within 10000

IV. RESULTS

A. Quantitative Results

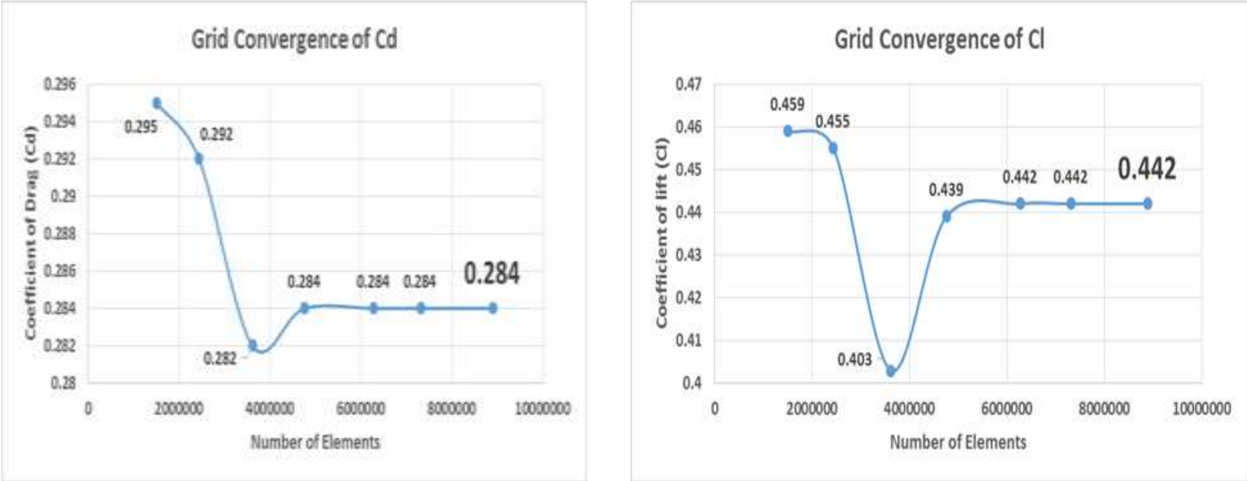

Figure 6Convergence of Drag and Lift Coefficients indicating grid convergence

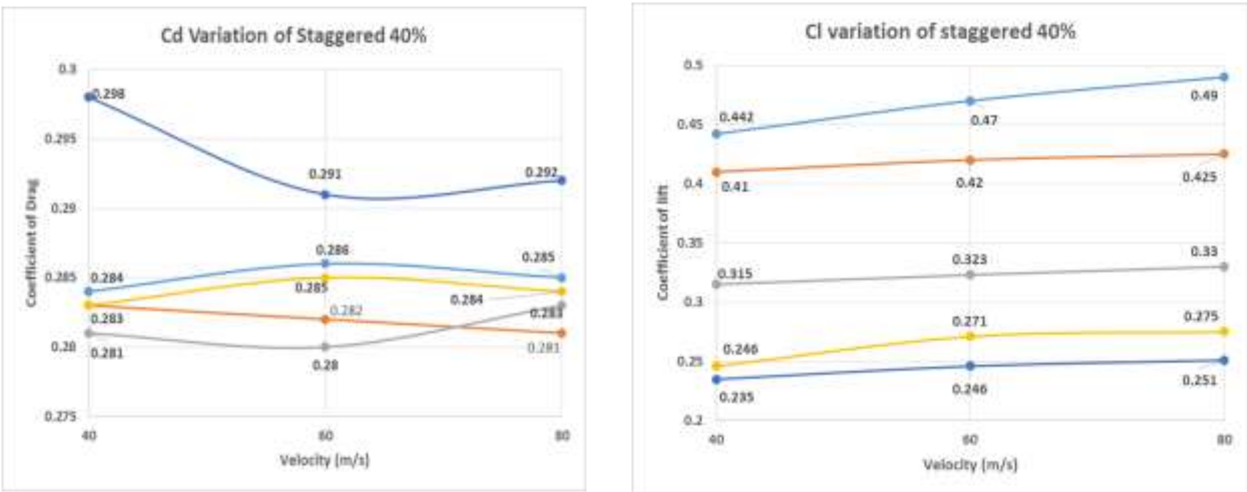

Figure 7. Drag and Lift performance at different forward velocities for the staggered arrangement with coverage ratio of 40%

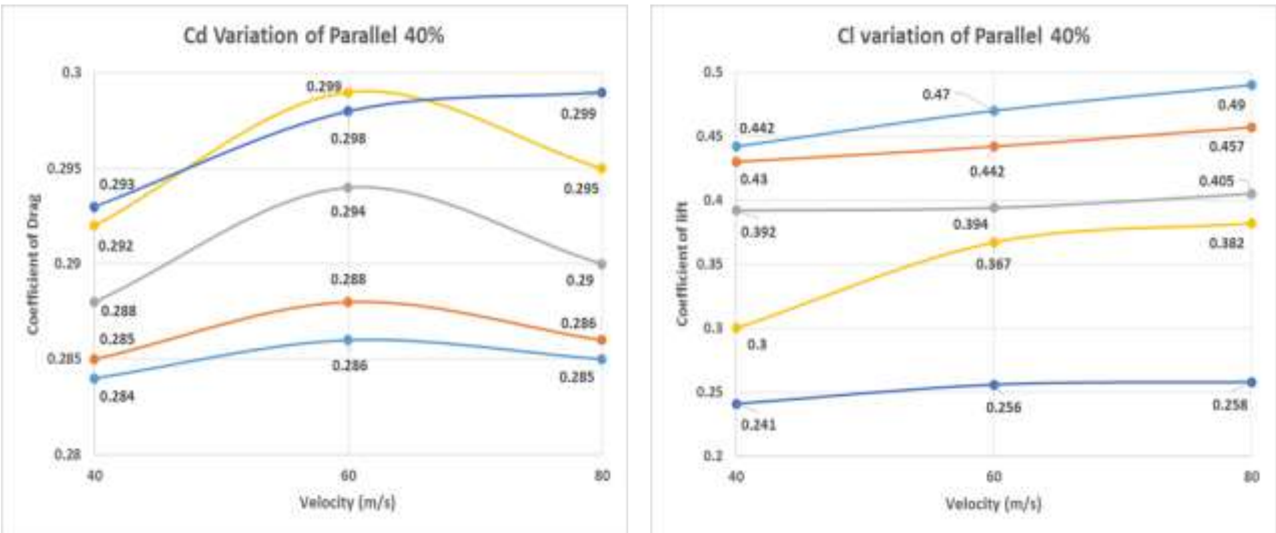

Figure 8. Drag and Lift performance at different forward velocities for the Parallel arrangement with coverage ratio of 40%

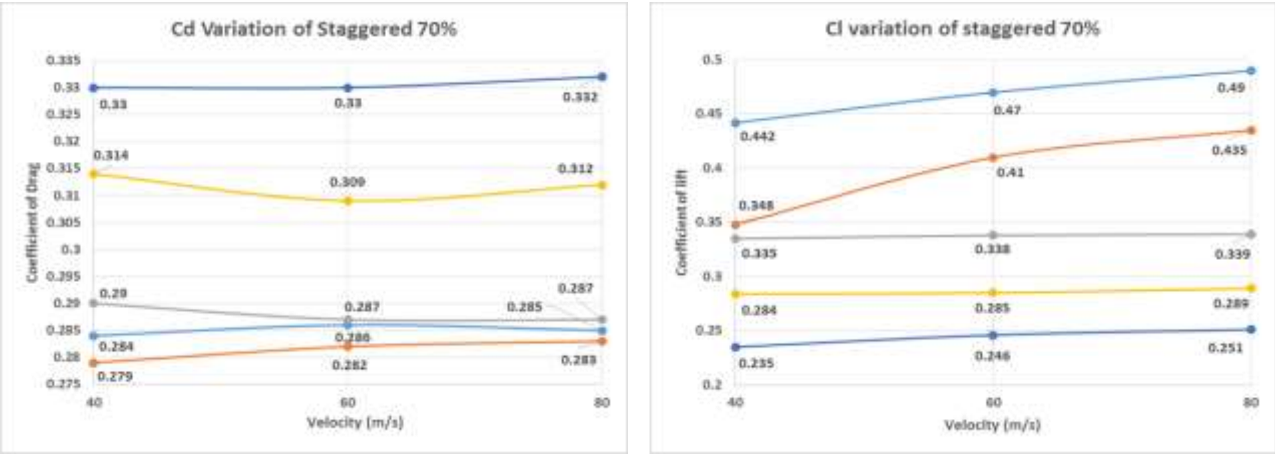

Figure 9. Drag and Lift performance at different forward velocities for the staggered arrangement with coverage ratio of 70%

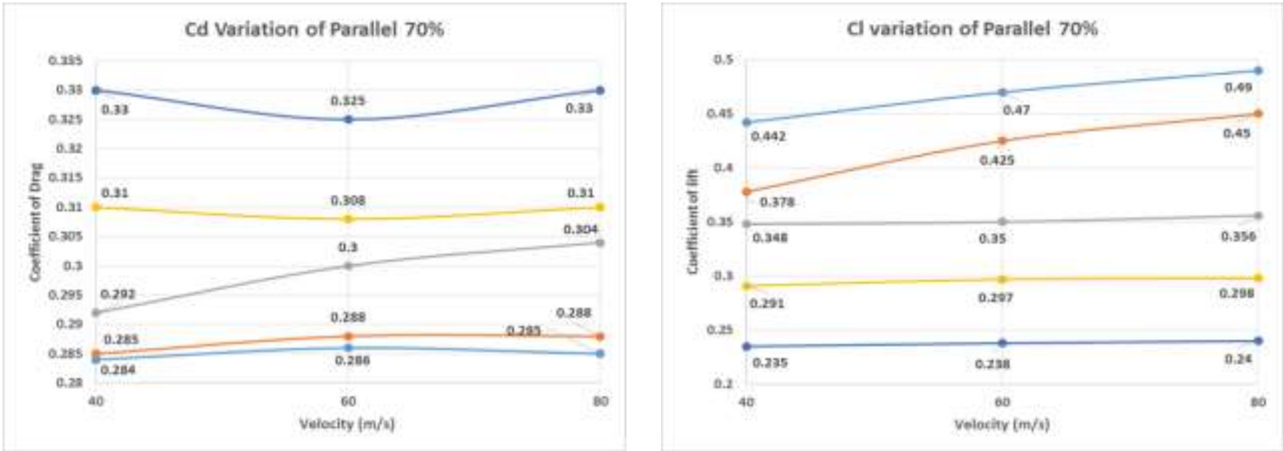

Figure 10. Drag and Lift performance at different forward velocities for the parallel arrangement with coverage ratio of 70%

B. Qualitative Results

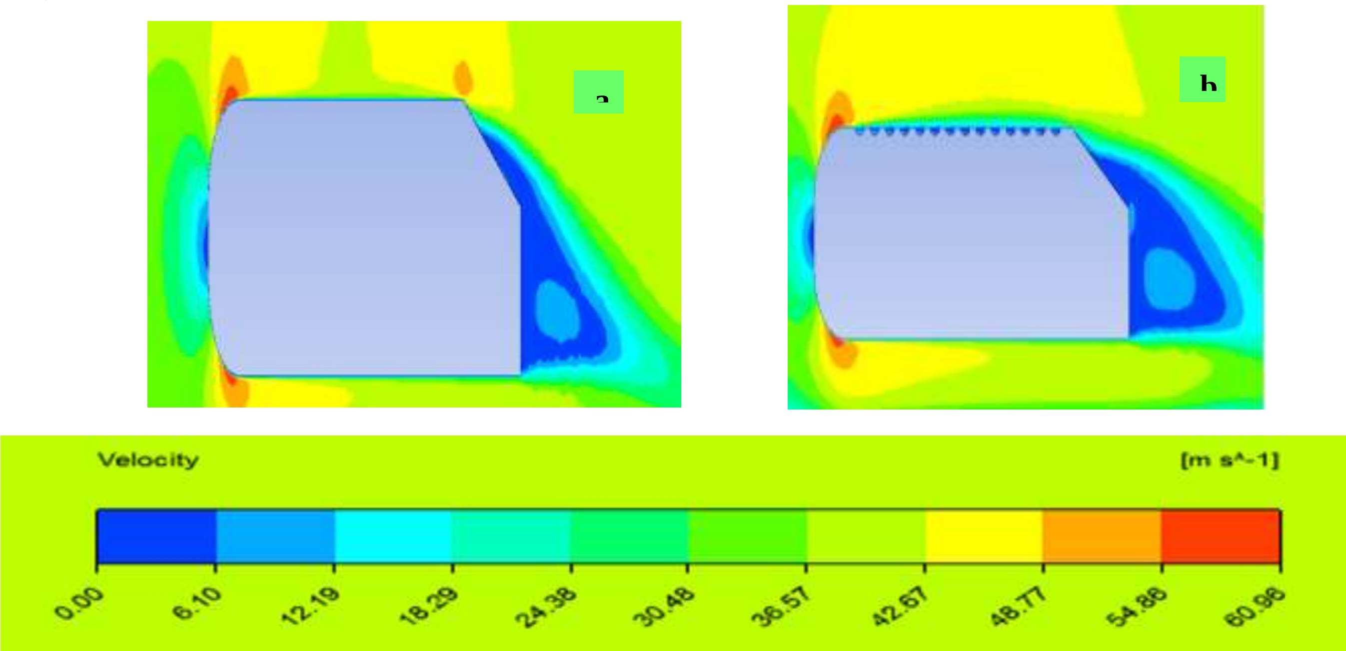

Figure 11. (a) The contours of velocity over the base model at 40m/s (b) Contours of velocity in the case of staggered dimples with DR30% and CR 40% at 40m/s

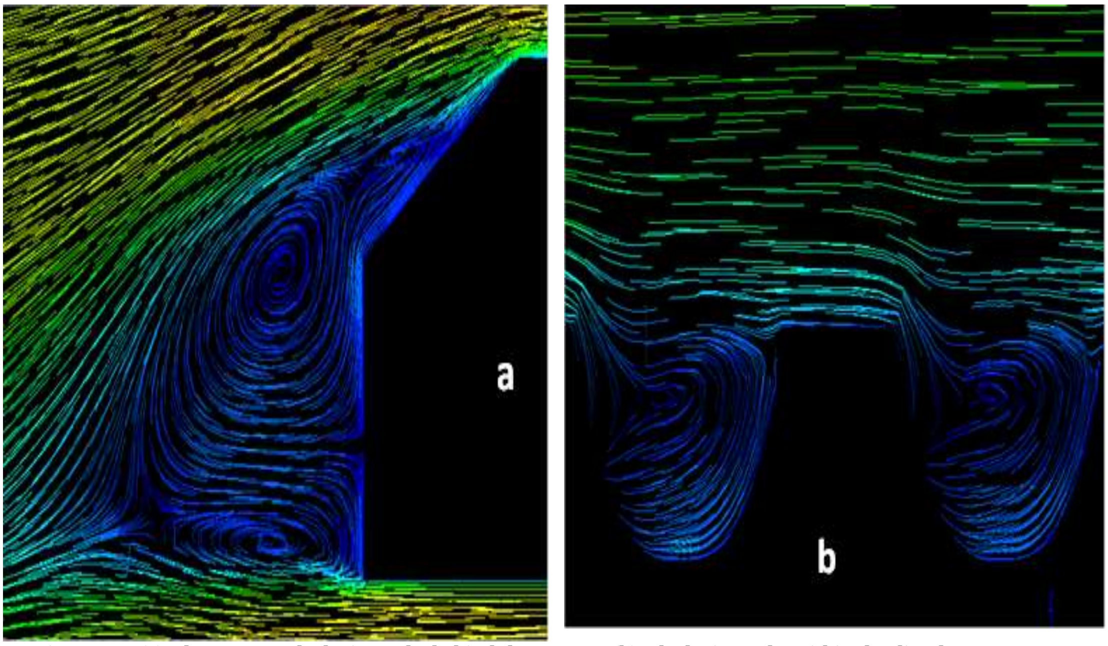

Figure: 12. (a) The counter-clockwise wake behind the car (b) Clockwise wake within the dimples

V. DISCUSSION

A. *General Trend and Justification for Drag Variation*

It can be observed from figure 7-10 that the drag variation of different dimple ratios showed great dependence on the arrangement and coverage ratio. When four dimple ratios (5%, 10%, 20% and 30%) were tested, the deepest dimple (DR 30%) consistently showed an increase in drag, irrespective of the arrangement and coverage ratio. On the other hand, the shallowest dimple (DR 5%) showed consistency in reducing drag across all the velocity ranges and the coverage ratios only in the staggered arrangement. The other two dimple ratios i.e., 10% and 20% showed drag reduction in a staggered arrangement with a coverage ratio of 40% but showed a drag increase in the staggered arrangement with a coverage ratio of 70%. However, all the dimple ratios tend to increase drag when the arrangement is parallel, irrespective of the coverage ratio.

At higher dimple ratios, flow separation was observed within the deep dimple pockets creating a local wake region. These local wakes are oriented against the direction of flow (clockwise) unlike the large wake region behind the car (anticlockwise) as shown in figure 12. These cross stream vortices can be held responsible for earlier flow separation and subsequent increase in drag on the rear slant in the cases with deep dimples. The reason for drag reduction in shallow dimples (such as DR 5%) can also be attributed to the absence of any such cross stream vortices. The dimples are too shallow to cause any adverse pressure gradients and to initiate any subsequent flow separation within the dimples. Hence the improvement in drag characteristic is analogous to the effects of increasing the surface roughness and the reduction of drag is very small. Thus the shallow dimples tend to decrease the friction drag by imparting some surface roughness onto the roof.

B. *General Trend and Justification for Lift Variation*

One of the most successful findings of this research is that all the dimple ratios consistently showed a reduction in lift performance (i.e., increase in downforce), irrespective of the depths. The data labels on the figure 7-10 show that the lift coefficient with dimples is less at higher vehicle speeds i.e., the wheels adhere to the road better at high speeds, which is the favourable condition for high-performance cars.

As the vehicle speed increased, the lift coefficient increased for the base model. This is intuitive since, with the increase in vehicle speed, the airflow under the vehicle would increase and tend to lift the wheels off the ground. However, with the use of dimples, the lift coefficient decreased with increase in dimple depth due to the cross stream vortices formed within the dimples. These local vortices when observed three-dimensionally have a force component acting downwards on the roof. This apparently pushes the vehicle downwards thus increasing the downforce. The fact that the decrease in lift coefficient is extremely low for the shallowest dimple due to the absence of cross-stream vortices, reiterates the importance of the local vortices within the dimples in lift reduction.

C. *Most Favourable Design*

Amongst all the dimple arrangements and ratios tested, the staggered arrangement with dimple ratio 10% and coverage ratio 10% tends to offer the most favourable results. The drag reduces up to 1.05% at 40m/s, up to 2.1% at 60m/s and 1.4% at 80m/s. The lift force decreased by 28.73% at 40m/s, 28.08% at 60m/s and 32.65% at 80m/s. This is the only arrangement which showed a reduction in both drags and lift coefficients.

However, if the application of this concept is on a high-performance car and demands a greater downforce even at the expense of an increased drag, the staggered arrangement with dimple ration 30% and coverage ratio 40% offers the best results. The lift coefficient reduced by 46.83% at 40m/s, 47.65% at 60m/s and 48.77% at 80m/s through the drag coefficient increased by 4.92% at 40m/s, 1.75% at 60m/s and 2.45% at 80m/s.

D. *Validation of Results*

The coefficient of drag of the base model was found to be 0.284 at 40m/s while the experimental results by *Ahmed S.R (1984)* [23] claims the value to be 0.298. This 5% variation is a classic example of the difference between CFD and experimental results. Across all dimple arrangements and coverage ratios, the drag ratio was found to increase with dimple depth. It was reported in this research that the shallowest dimple showed drag reduction and similar observations were made by *Isaev .S.A (2010)* [19], *Alekseev V.V et al (1998)* [11] and *Chowdhury .H et al (2016)* [7]. It was observed that if at all there is any drag reduction, it is possible only with the staggered arrangement which was similar to the conclusions of *(Olaf) van Campenhout et al (2016)* [12]. The cross stream vortices in dimples were found to be the most important reason for the drag increase in this research. *Won .S.Y et al (2005)* [17], *Kovelenko .G.V et al* (2010) [10] and *Ligrani .P.M et al* (2005) [16] also drew similar conclusions. It was observed that friction drag component decreased which was similar to the conclusions of *Chen .Y et al (2012)* [15], *Burgees .N.K et al (2005)* [14] and *Ligrani .P.M et al (2005)* [16]. In this research, the maximum drag reduction that was achieved at 40m/s was 1.76%. Chear and Dol (2015) [13] reported a 1.95% reduction, which validates the findings in this research.

CONCLUSIONS

A wide range of dimple ratios, their arrangements and sizes have been tested at various vehicle speeds, covering almost every variable parameter, thus making the observations extensive and holistic. Unlike many previous works conducted in this domain, the lift performance of the dimples has been critically evaluated.

Like most aerodynamic devices, dimples on roofs had a significant variation in both drags and lift the performance of an automotive. It has been identified that all the dimple arrangements tend to increase the downforce at high speed which is highly favorable for high-performance cars. Shallow dimples tend to decrease drag and the cross stream vortices within the dimples were found to be the reason for increased drag and decreased lift (increased downforce). Amongst many dimple ratios tested there was only one arrangement which offered a considerable reduction in both drag and lifts coefficients. Though a wide spectrum of dimple arrangements and ratios has been taken into consideration, the choice will be dictated by its application (a type of vehicle) and the designer’s discretion.

The biggest advantage of the suggested modification in that no additional weight is added to the vehicle. Modern high-performance cars have additional devices which reduce lift such as the wings and spoilers which apparently increase the weight of the vehicle. However, if roof dimples and such lift reducing devices work in synergism, the overall lift reduction would be much higher. Though roof dimples are not feasible in Formula 1 (where there is no roof by itself), the effect can be experienced in drag racers such as the cars used in NASCAR Sprint, where cornering performances are of prime importance.

By 2050, the world would be dominated by autonomous cars and high-speed road travel is soon going to be a reality. In such an automated environment, the looks and designs of cars would move towards more streamlining and efforts would be taken to make the cars more aerodynamically efficient. Hence for faster travels even on passenger cars, the downforce is of paramount importance and the solution offered in this research work has a remarkable potential in the near future.

ACKNOWLEDGMENT

I am always grateful to the love and support of my parents and dedicate the successful completion of this research to the two noble souls. I also extend my gratitude to my teachers and friends for their valuable guidance and continued support.

REFERENCES

[1] Wolf –Heinrich Hucho, *Aerodynamics of Road Vehicles (4th edition),* SAE International, ISBN 0-7680-0029-7, USA, 1998.

[2] Sykes .D.M and Scibor-Rylski zA.J. *Road Vehicle Aerodynamics (2nd edition),* Pentech Press Limited, ISBN 0-7279-1805-5, Great Britain, 1984.

[3] Aoki .K, Muto.K, Okanaga .H, “Aerodynamic Characteristics and Flow Pattern of a Golf Ball with Rotation”, *8th Conference of the International Sports Engineering Association (ISEA).* Science direct, Procedia Engineering 2 (2010) 2431-2436, 2010.

[4] Versteeg H. K and Malalasekara .W, *An Introduction to Computational Fluid Dynamic, the Finite Volume Method,* Longman Scientific and Technical, ISBN 0-582-21884-5, England, 1995.

[5] Bansal .R. K, *A Textbook of Fluid Mechanics and Hydraulic Machines,* Lakshmi Publications, India, 2011.

[6] Mehta .R.D and Jani Macari Pallis, “Sports Ball Aerodynamics: Effects of Velocity, Spin and Surface Roughness”, *Materials and Science in Sports*, pp 185-197, Warrendale, 2001.

[7] Chowdhury .H, Loganathan .B, Wang .Y, Mustary .I, Alam .F, “A study of Dimple Characteristics on Golf Ball Drag”, *11th Conference of the International Sports Engineering Association (ISAEA),* Elsevier, Australia, 2016.

[8] Kray .T, Franke .J, Frank .W, “Positive and negative Magnus Effect on a Rotating Soccer Ball at high Reynolds Number”, *Aachen University of Applied Science,* Germany, 2012.

[9] Tay C. M. J, Khoo B. C, Chew Y. T, “Mechanics of Drag Reduction by Shallow Dimples in Channel Flow”, *Physics of Fluids 27,035109,* Singapore, 2015.

[10] Kovalenko G. V, Terekhov V. I, Khalatov A. A, “Flow Regimes in a Single Dimple om the Channel Surface”, *Journal of Applied Mechanics and Technical Physics,* 51, pp 839-848, 2010.

[11] Alekseev V. V, Gachechiladze I. A, Kiknadze G. I, Oleinikov V. G, “Tornado-like energy transfer on three-dimensional concavities of reliefs – structures of self-organizing flows , their visualization and surface streamlining mechanisms”, *Transactions of 2nd Russian National Conference on Heat Transfer Intensification Radiation and Complex Heat Transfer,* Vol.6, pp 33-42, 1998.

[12] O.W.G. (Olaf) van Campenhout, M.(Michiel) van Nesselrooji, L.L.M.(Leo) Veldhuis, B.W.(Bas) van Oudheusden, F.F.J. (Ferry) Schrijer, “Flow visualization over drag reducing dimpled surface in turbulent boundary layers using particle image velocimetry”, *18th International Symposium on the Application of Laser and Imaging Technologies to Fluid Mechanics,* Portugal, 2016.

[13] Cheer C.K and Dol S.S, “Vehicle Aerodynamics: Drag Reduction by Surface Dimples”, *International Journal of Mechanical, Aerospace, Industrial, Mechatronic and Manufacturing Engineering,* Vol.9 No.1, 2015.

[14] Burgees N. K and Ligrani P. M, “Effects of dimple depth on Channel Nusselt number and friction factors”, *Journal of Heat Transfer, 127(8),* pp. 839-847, 2005.

[15] Chen.Y, Chew.T.Y, Khoo.B.C, “Enhancement of Heat Transfer in Turbulent Channel Flow over Dimpled Surface”, *International Journal of Heat and Mass Transfer,* Elsevier, Singapore, 2012.

[16] Ligrani .P.M, Burgees.N, Won .S.Y, “Nusselt number and flow structure on and above a shallow dimpled surface within a channel including effects of inlet turbulent intensity level”, *The American Society of Mechanical, Journal of Turbomachinery,* 127, pp 321-330, 2005.

[17] Won.S.Y, Zhang.Q, Ligrani .P.M, “Comparison of flow structures above dimpled surfaces with different dimple depths in a channel”, *Physics of Fluids,* 17,045105, 2005.

[18] Lienhart.H, Breuer.M, Koksay.C, “An experimental/numerical investigation on drag reduction by dimples”, *PAMM.Proc.Appl, Math,* 10, pp 31-34, 2010.

[19] Isaev.S.A, Kornev.N.V, Leontiev.I, Hassel.E, “Influence of Reynolds Number and the spherical dimple depth on turbulent heat transfer and hydraulic loss in a narrow channel”, *International Journal of Heat and Mass Transfer,* pp178-197, 2010.

[20] Gilkeson .C.A, Thompson .H.M, Wilson.M.C.T, Gaskell .P.H., “Quantifying passive ventilation with small livestock trailers using computational fluid dynamics”, *Researchgate,* UK, 2016.

[21] Landge.A.A and Palande.D.D, “Aerodynamics Study of automobile car Ahmed Body using CFD Simulation to predict the drag coefficients and downforces”, *International Engineering Research Journal (IERJ),* ISSN 2395-1621, 2016.

[22] Ahmed.S.R, “Influence of base slant on the wake structure and drag of road vehicles”, *Transactions of the ASME, Journal of Fluids Engineering,* Vol.105 pp 429-434, 1984.

[23] Mohammadi .B and Pironneau .O, *Analysis of K-epsilon turbulence model,* Research in Applied Mathematics, ISBN 2-225-84391-0, Paris, 1994.

,

