iJRASET
International Journal For Research in
Applied Science and Engineering Technology

# INTERNATIONAL JOURNAL FOR RESEARCH
IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: CAAA-2018 Issue: onference Month of publication: April 2018
DOI:

# Numerical Investigation on Aerodynamic Drag Reduction of a Two-Seater Car Using Dimples and Vortex Generators

Mr. Gokul Raj S[1], Mr. Karthikeyan J[2], Mr. Ajith A[3]
[1]*PG Scholar,* [2]*Assistant Professor, Dept. of Aeronautical Engg. Nehru Institute of Engg. & Tech. Coimbatore, TamilNadu.*

***Abstract: In the recent trends of automotive aerodynamics a wide research works have been carried out to reduce drag produced by the road vehicles and hence to improve the aerodynamic performance of the automobiles. For most of the sports vehicles and passenger cars, aerodynamic drag directly influences the speed and fuel economy. The wake formed behind the passenger cars and sports utility vehicles along with a strong flow separation is the major cause for drag production in automobiles. Various techniques have been identified to minimize the drag by influencing the flow field at the rear end. This paper focuses on the drag reduction technique using various shapes of dimples. Also a comparison of dimples and vortex generators at the rear end has been done to justify better aerodynamic performance of passenger cars. In the present investigation, Computational Fluid Dynamics (CFD) is effectively used to simulate the flow field and comparison of aerodynamic performance. The Drag Co-efficient (CD) of the car model has been compared with and without dimples along with vortex generators***
***Keywords: Drag Co-efficient; Dimples, Vortex Generators; CFD (Computational Fluid Dynamics)***

## I. INTRODUCTION

Road transportation is an unavoidable means of transport and the engineering behind the transport vehicles always leads some research and development activities. Starting from the power train system to component level optimization of the road transport vehicles plays a crucial role in the fuel efficiency and economy of the vehicles. The traction force on the car is mainly influenced by the aerodynamic loads acting on it.

There are various forces acting on a car. The force of air that pushes the car straight down and directly affects the handling of a vehicle, adding traction in acceleration, braking and cornering is down-force. Increased vertical force on the tires (mainly caused by gravity), creates better tire grip and a better overall performance of your car. Down force is important because the grip depends on how hard the tires are being pushed down into the road.

Drag is the force of air in the direction opposite the car's motion, acting toward the rear and tends to slow the car. Drag exists on all vehicles and affects the handling of vehicle. Lift, in automobile terms, is the component of aerodynamic forces acting in a perpendicular direction to the flow of air over the vehicle.

Several active and passive methods of automobile wake reduction have been suggested, including the use of more aerodynamic shapes, powered suction, and boundary layer re-energization. The power required for suction to generate a noticeable change in drag is far greater than the capacity of the engine of the automobile. Thus, the powered suction technique is impractical.

To avoid long streamlined rear sections of vehicles it is necessary to re-energize the boundary layer. The boundary layer can be re-energized by mixing some of the free stream air with the boundary layer air.

This mixing increases the energy of the air in the boundary layer delaying boundary layer separation and the size of the wake produced by the vehicle. Some vehicles employ wing like aero foils as turning vanes to assist in the directing flow, thus reducing the wake region.

Similarly the vortex generator is a device that can be used to re-energize the boundary layer delaying the flow separation as shown below. Also the boundary layer can be influenced by the changing the surface parameters by introducing dimples.

## II. METHODOLOGY

### *A. Physical Model*

The domain considered for the present numerical analysis is shown in Figure 1. The domain length is considered 4 times of the total car length in the upstream region and 8 times in the downstream region. The overall length of the car is around 4400 mm.

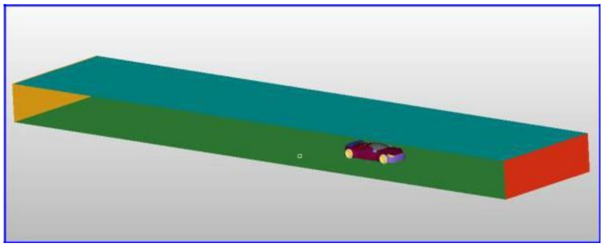


Figure 1. The physical model considered for the present investigation

Figure 2 shows the car model with dimples at the rear end of the windshield. The sports utility car model is of two seater with a track length of 2050 mm.

The rear end wind shield is provided with dimples of circular and hexagonal shapes of same size. Dimples are systematically arranged over the car model. The figure 3 shows the location of the dimples over the wind shield where it is arranged in a system of arrays to delay the flow separation which could happen at the rear section of the model.

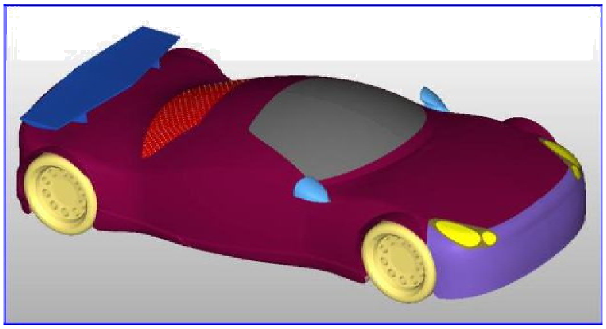


Figure 2. Wind Turbine Blade with Different Twist angle at Root Section of the Blade

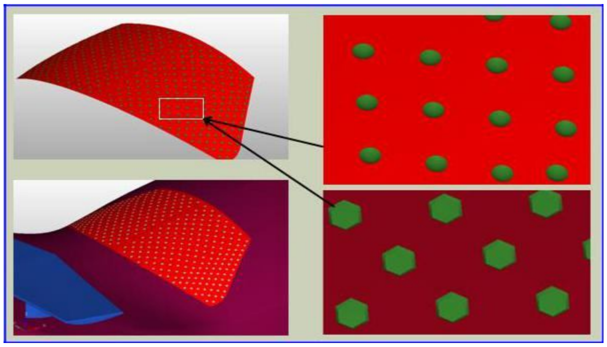


Figure 3. Location and Shapes of Dimples – Over Wind Shield

Figure 4 depicts the vortex generators (VGs) used for present numerical analysis. There are 9 VGs used in the rear and of the car just before the rear wind shield. The VGs are arranged with equal pitch on both sides.

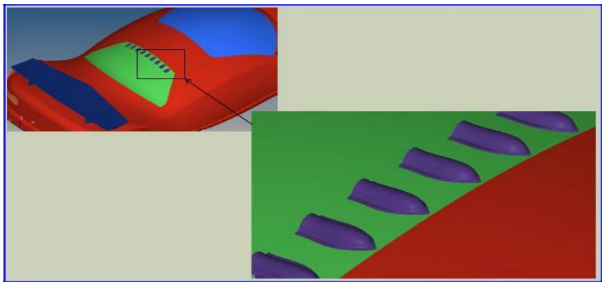


Figure 4. Location and Shapes of Vortex Generators

*B. Computational Model*

The computational domain used for the present CFD analysis is given in Figure 5. The surfaces are meshed with triangular elements and the minimum element size used for the analysis is 2mm and the maximum element size is 40mm at the bottom of the vehicle. Mesh refinements have been carried out at the regions of interest.

Figure 6 shows the mesh refinement near the vortex generators and dimples. The minimum element size used near the dimple is 0.5 mm and the profile of the vortex generators are captured properly.

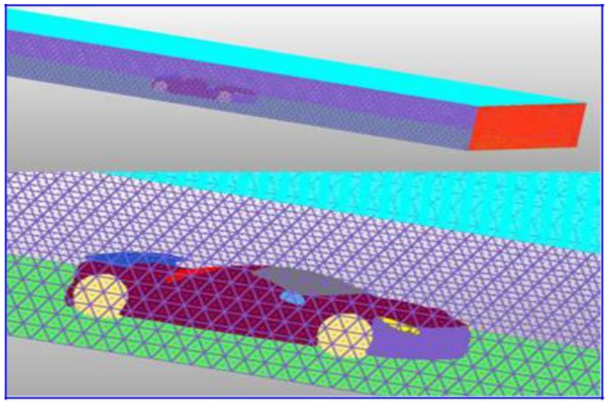


Figure 5. Computational Domain – Surface Mesh

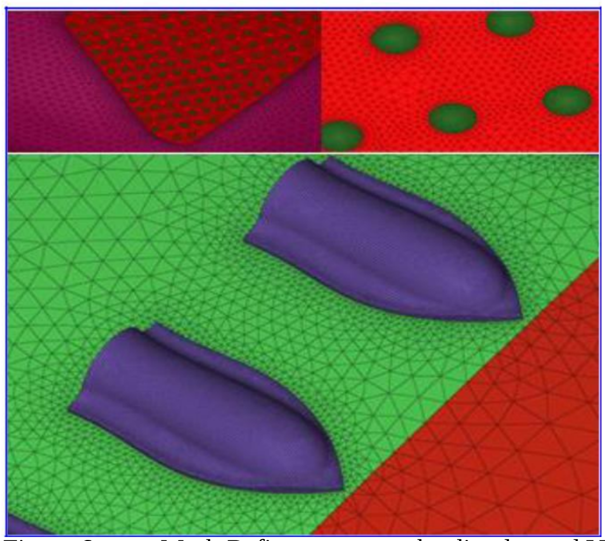


Figure 6. Mesh Refinement near the dimples and VGs

The cut sectional view of the volume mesh near the car model with tetra element growth is shown in Figure

7. The tetra element growth considered around the vehicle model is with a growth ratio of 1.4.

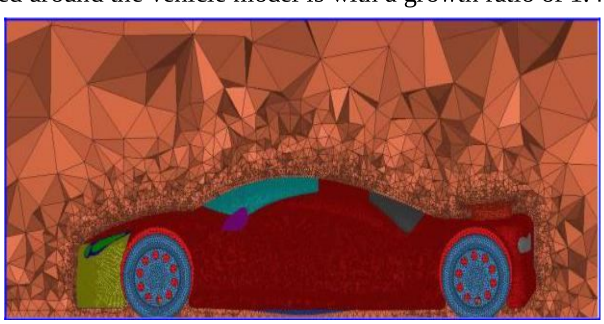


Figure 7.BVolume Mesh Cut Sectional View

*C. Physics Definitions*

The discretized domain is given with appropriate boundary conditions in Ansys Fluent. The inlet velocity is given with 55.5555 m/s (100 Kmph) and outlet is given with a pressure outlet boundary conditions for which the gauge pressure is set as 0 Pa. The fluid considered for the present analysis is atmospheric air with P= 1 Atm and T = 300 K.

The turbulence model used for the present analysis is RNG k – Epsilon model. The analysis is carried out with steady and incompressible assumptions. The entire physics definition is given in Table 1.

TABLE I. BOUNDARY CONDITIONS

| Detail | Condition | Value |
|---|---|---|
| Inlet | Velocity Inlet | 55.55 m/s |
| Outlet | Pressure Outlet | 0 Pa (gauge) |
| Sports Car Body, Spoiler, Wheels, etc., | Wall Boundary | No Slip Wall |
| Reference | Ambient | 101325 Pa |
| Turbulence | RNG k-Epsilon | 5% Turbulence Intensity |
| Solution Algorithm | SIMPLE | Steady and Incompressible |
| Fluid | Atmospheric Air | Density = 1.225 kg/m$^3$ Viscosity = 1.79e-05 kg/m-s |
| Equations Solved | Flow and Turbulence | Continuity X- Momentum Y- Momentum Z- Momentum K and Epsilon |

## III. RESULTS *AND* DISCUSSIONS

*A. Effect of Dimple Shapes*

The computational simulation of the car model with and without dimples is carried out and the results are post processed after the solution convergence. The convergence criteria for all equations are in the order of 1*E-04. The presence of dimples has a significant influence on the rear end flow field of the car. The flow separation behind the car model is greatly influenced by the location and series of dimples on the rear wind shield. Figure 8 shows the effect of dimples when compared with car model without dimples.

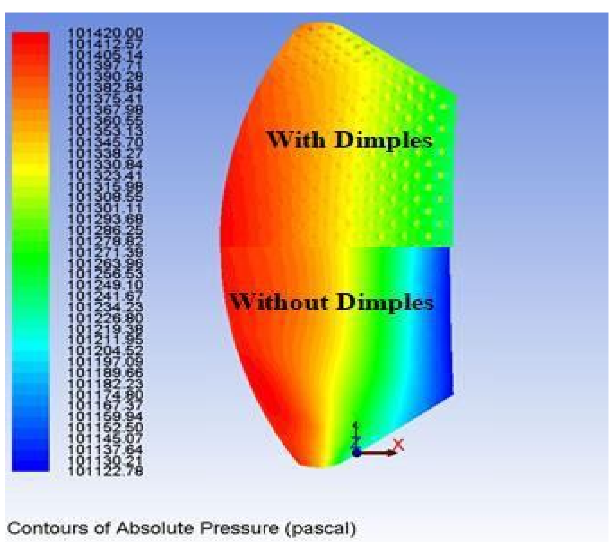



Figure 8. Rear Wind Shield – Static Pressure Distribution – with and without dimples

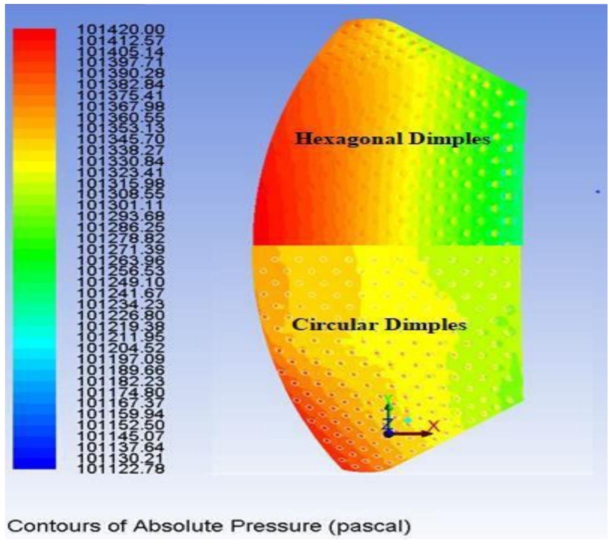



Figure 9. Rear Wind Shield – Static Pressure Distribution –Comparison of Hexagonal and Circular Dimples

Figure 9 shows the effect of dimple shapes on the static pressure distribution of rear wind shield. The flow separation is delayed further due to influence of the dimples.

This is clearly depicted in Figure 9. Also the static pressure shift is clearly shown in Figure 10 due to presence of hexagonal dimples

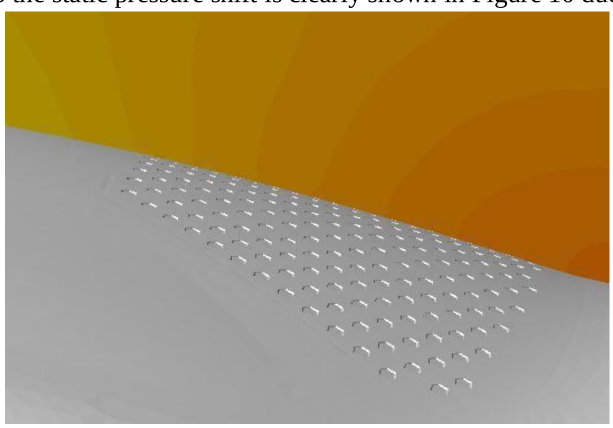


Figure 10. Static Pressure Variations – Presence of VGs

*B. Effect of Vortex Generators*

The flow field behind the car model is also influenced by the presence of add-on devices like vortex generators and spoilers. In the present numerical investigation a new design of vortex generators are analyzed along with spoiler. Figure 11 shows the effect of vortex generators.

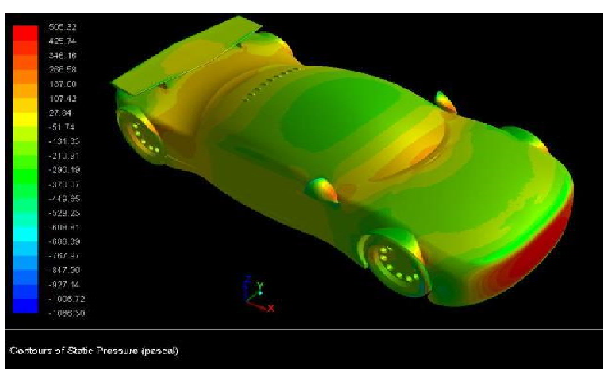



Figure 11. Static Pressure Variations – Presence of VGs

Figure 12 depicts the turbulent intensity variations at the rear side of the car due to presence of VGs.`

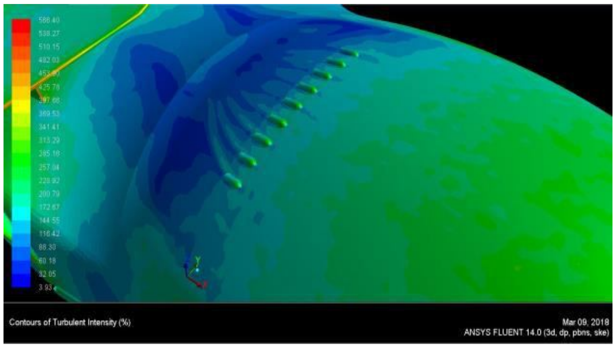



Figure 12. Effect of VGs- Turbulent Intensity

*C. Comparison of Dimples with Vortex Generators*

The overall drag force comparison is shown in the bar-chart given below. From the CFD results it is observed that the drag force is significantly reduced due to the presence of dimples and vortex generators. The non-dimensional parameter of drag (Drag Co-efficient) is shown in Figure 13.

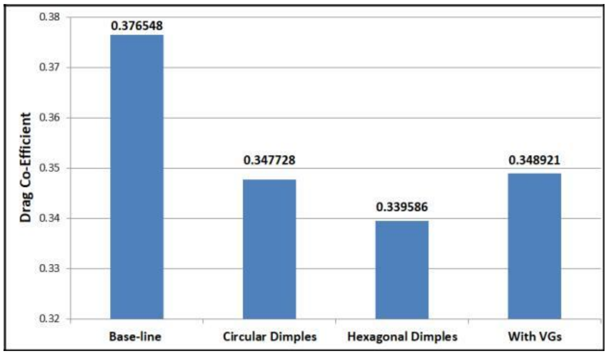



Figure 13. Comparison of Drag Co-Efficient

## IV. CONCLUSION

The computational simulation is able to predict the flow patterns and wake behind the car model accurately and the commercial CFD codes can be used to optimize the car body aerodynamics. From the present numerical investigation it can be concluded that the dimples are more effective in reducing the drag forces along with VGs and there is a significant reduction in drag co-efficient. The flow separation is reasonably delayed at the rear end which in turn helps to reduce the drag force.

## REFERENCES

[1] Firoz Alam, Tom Steiner, Harun Chowdhury, Hazim Moria, and Iftekhar Khan, “A study of golf ball aerodynamic drag” 5th Asia-Pacific Congress on Sports Technology (APCST), Elsevier, ScienceDirect, Procedia Engineering 13 (2011) 226–231.

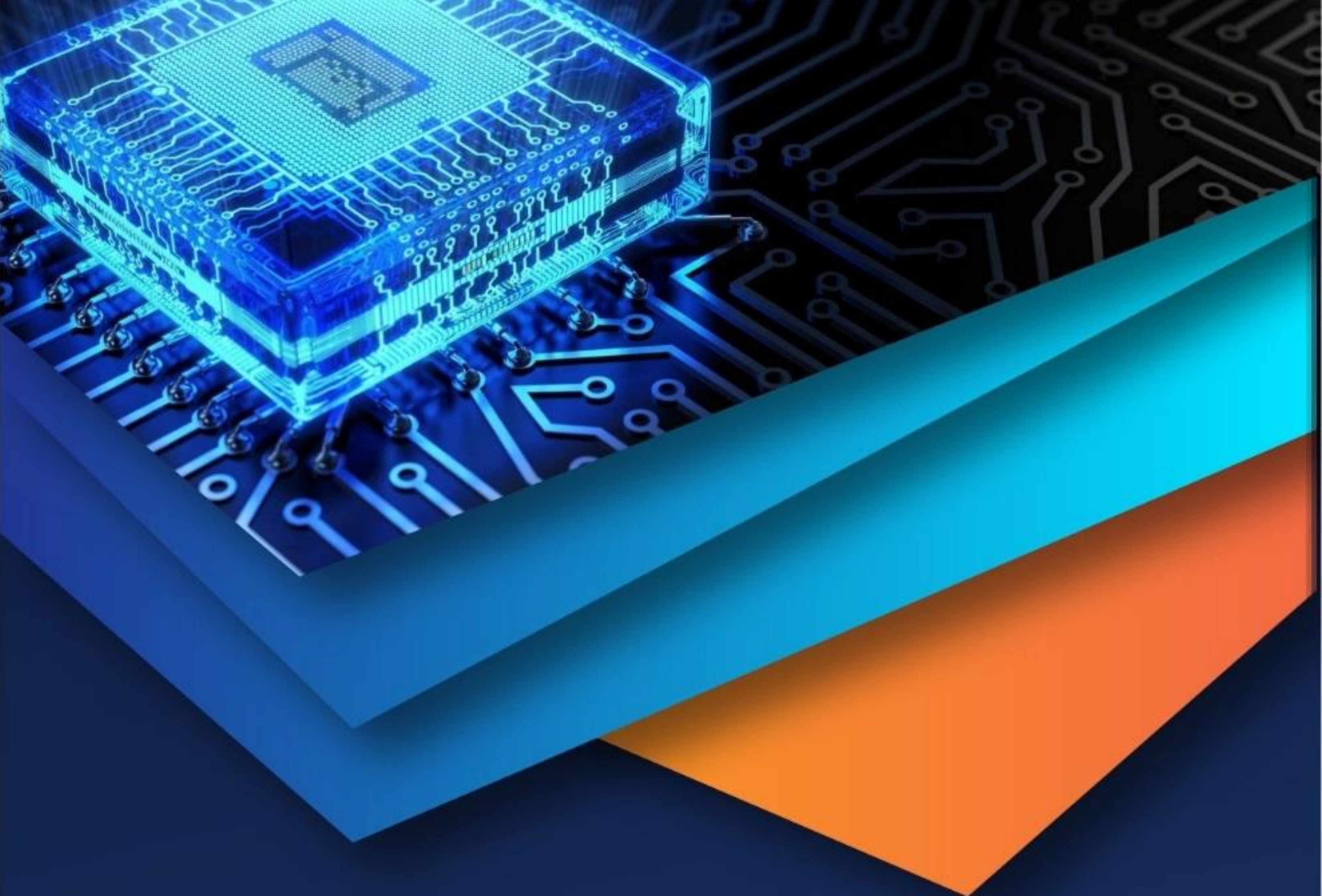
[2] Choi J, Jeon W, Choi H. “Mechanism of drag reduction by dimples on a sphere”, Physics of Fluids 2006;18:1-4.

[3] Ting LL. Effects of dimple size and depth on golf ball aerodynamic performance. In: 4th ASME-JSME Joint Fluids Engineering Conference 2004.

[4] Bryton A. Horner, “Fuel Troubles: Improving Car Aerodynamics and Fuel Efficiency through Dimples”, CALIFORNIA STATE SCIENCE FAIR.

[5] Sai Sujith and Ravindra Reddy (2012) “CFD analysis of Sedan Car with vortex generators” – IJMEAR 2012, Vol 3.




10.22214/IJRASET

INDEX COPERNICUS 45.98

IMPACT FACTOR: 7.129

IMPACT FACTOR: 7.429


# INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY