

Numerical Analysis of Drag Reduction System in Automobile

Bibhuti Bhusan Samantaray*, Chinmay Kumar Mohanta**

Abstract

Aerodynamic drag plays a very vital role in speed and stability of the automobile. Too much drag reduces the speed and efficiency while too little drag creates very much instability in the vehicle. DRS or drag reduction system is a very useful tool in formula one motoracing where they use it to reduce drag & gains speed advantage. Hence this article is concentrated on analysis of drag reduction with different speed of the formula one car numerically. A rear wing model which is a very vital part of the DRS is designed using CATIA Vr520 software and simulating analysis is made using ANSYS 14.0 software. With a mesh relevance of +30000, it is found that with the application of DRS and acrophobic layer, the air resistance is reduced by 26 %. This reduction enhances the efficiency as well as performance of vehicles.

Keywords: Automobile, Drag Reduction, Efficiency, Resistance

INTRODUCTION

Many researchers have experimented to reduce drag and find the reason behind them. The car aerodynamics was properly studied along with understanding the streamline designing to increase downward force. [1] Various computational analysis is done in automobile sector using a simplified car body [2] starting from 1st generation to f1 generation [3] in order to decrease the drag. [4] Numerical and experimental study was done to calculate drag only for sedan and hatchback cars but no drag reduction system was experimented. [5]

Vortex generators are fitted to prevent vortices forming to the diffuser plates mounted low at back. [6] The effectiveness of vortex generator is only restricted by the body shape of the car. Numerous drag reduction technique were discussed concerning sport utility car and racing car but were not very informative. [7] Studies on race car aerodynamic drag include both 2-Dimensional and 3-Dimensional analysis but the drag reduction is done by the spoiler only. [8] Very limited paper have given clear indication that which specific area should be used in drag calculation as different area is subjected to different drag force. In this paper

a rear wing is designed for sedan and sports utility vehicles taking inspiration from McLaren MP4-29. [9] The exploitation of ground effect downward force played an important role in understanding drag reduction techniques. To reduce drag, designers built lower and wider wings. Wings are used to create vortices at their edges. It is a rotating fluid that creates a low pressure zone at the center. Creating vortices lowers the overall local pressure of the air. Barge boards and vortex generators are also mounted low at the back.

Then came the movable aerodynamic idea with controlled system from cockpit. With further modification, in 2011, Drag Reduction System came into limelight with driver adjustable electronically governed rear wing system. DRS stalls the rear wing of car by opening a flap which leaves a 50 mm horizontal gap in the wing, thus reducing drag and allowing higher speed with lesser air resistance. In this work numerical simulation is performed to analyze the drag magnitude with DRS and without DRS. Various speed value is simulated in ANSYS 14.0 software to verify the reduction in air resistance.

* Department of Mechanical Engineering, RCET, Bilai, Chhattisgarh, India. Email: bbsr.comofficial@gmail.com

** Department of Mechanical Engineering, Gandhi Engineering College, Bhubaneswar, Odisha, India.
Email: Chinmaymohanta551@gmail.com

NUMERICAL MODELLING

Usually the rear wing is a single endplate which tilts in a definite angle to generate downward force. Here more wing angle and surface creates more down force which in turn reduces speed. But here the rear wing is splitted into two set of aerofoil which is connected to each other. The upper aerofoil is one large single element whereas the lower one is responsible for creating down force. The lower wing when tilted to a certain angle which creates a low pressure region to create more down force below the car.

The position of two wings should be relative to each other as if they were very close then the resultant force will be in opposite direction and thus cancel each other. Two wings produce more down force as the lift coefficient increases and the drag decreases.

Table 1. Calculated Value of Air Resistance

	Velocity (m/s)		
	88.88	83.33	77.77
With DRS (N/m)	17.291	16.211	15.132
Without DRS (N/m)	20.171	20.171	20.171
	18.912	18.912	18.912
	17.651	17.651	17.651

DETERMINATION OF AIR RESISTANCE

Air resistance is the resistance offered by air to the movement of a vehicle. The air resistance has an influence on the performance, ride and stability of the vehicle and depends upon the size and shape of the body of the vehicle, its speed and the wind velocity. The last term should be taken into account when indicated, otherwise it can be neglected. Hence in general, air resistance is calculated as shown [10],

$$R_a = K_a A V^2$$

A = Projected Frontal Area, m²

V = Speed of the Vehicle, km/hr

K_a = Coefficient of Air Resistance, Nhr²/m²km²

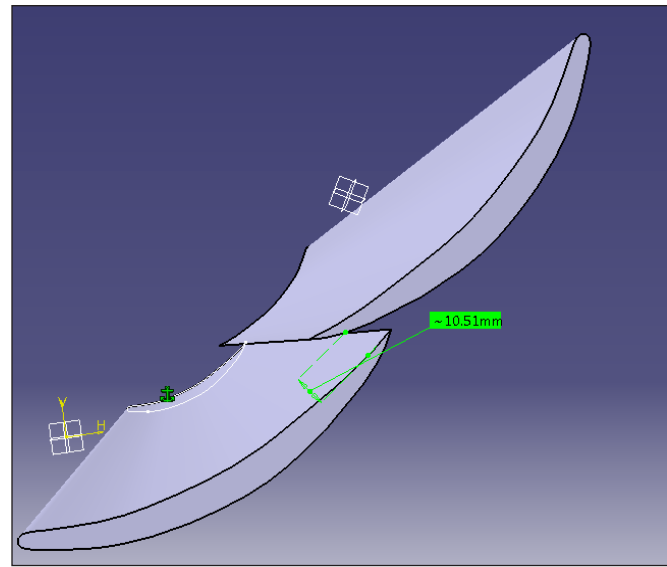


Fig 1. Wings in Closed Position.

The rear wing is designed by using the modelling software CATIA Vr5. In modelling the time is spent in producing the complex 3-Dimensional models and the risk involved in design and manufacturing process is minimised. While designing, a gap of 10.51 mm is left between both end edges of the wings which shows the wings are in closed position i.e. DRS is Off (fig 1).

Similarly another wing is designed having a gap of 50 mm between both end edges which shows the wings are in opened position i.e. DRS is On (fig 2). The wing length is kept constant i.e. 1650 mm. Later both the wings are imported to ANSYS software for analysis work. ANSYS 14.0 software has the latest CFD used for simulating the air flow. The file is imported from CATIA to ANSYS by saving it in .igs format. The imported model is meshed having a mesh relevance of +30,000. After constraining the meshed model, it is subjected to boundary conditions and an air pocket is created to simulate the air flow inside it. Air velocity of 320 km/hr, 300 km/hr and 280 km/hr are striked over both the designed wing surface.

A solver mode in ANSYS calculates the result without manual

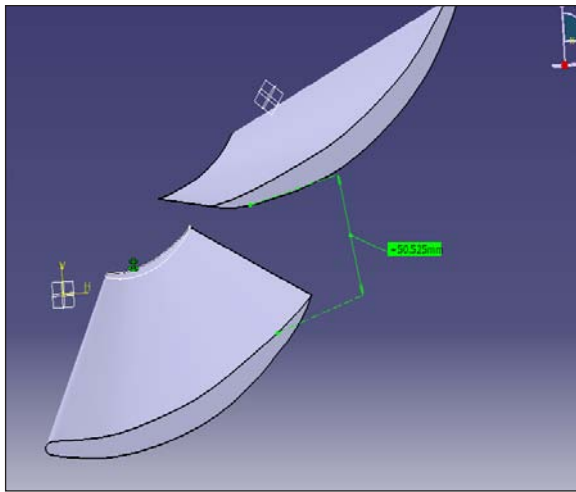


Fig 2. Wings in open position

Intervention reducing the time as compared to a human. The static analysis is done and their relative graphical results have been observed.

RESULT AND DISCUSSION

Effect of air resistance

The term drag area is derived from aerodynamics, where it is the product of some reference area (such as cross-sectional area, total surface area, or similar) and the drag coefficient. The air resistance formula is used here with surface area of the wing as 0.5775 m^2 for closed wings and 0.495 m^2 for the opened wing. The velocity unit is converted into m/s and the calculation is tabulated as shown.

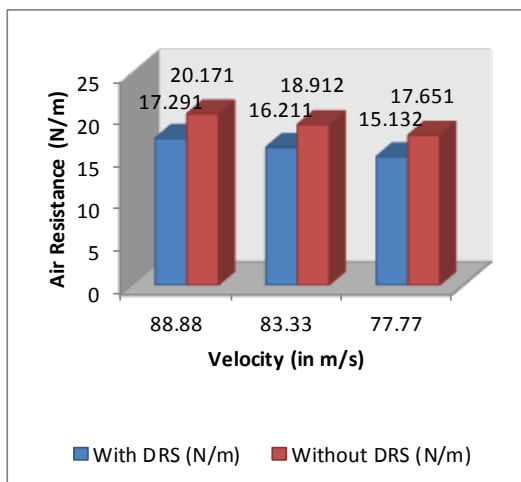


Fig 3. Graph Showing Air Resistance with Velocity

ANSYS SOLVER RESULTS

Different air velocity are simulated on the wings and the velocity streamline gradient is shown. With increase in velocity the streamline gradient also increase which shows the effect of air resistance increase. The fig 4, 5, 6 shows the wings are in closed position i.e they act as a single element wing. Hence it is said that Drag Reduction System is in off mode.

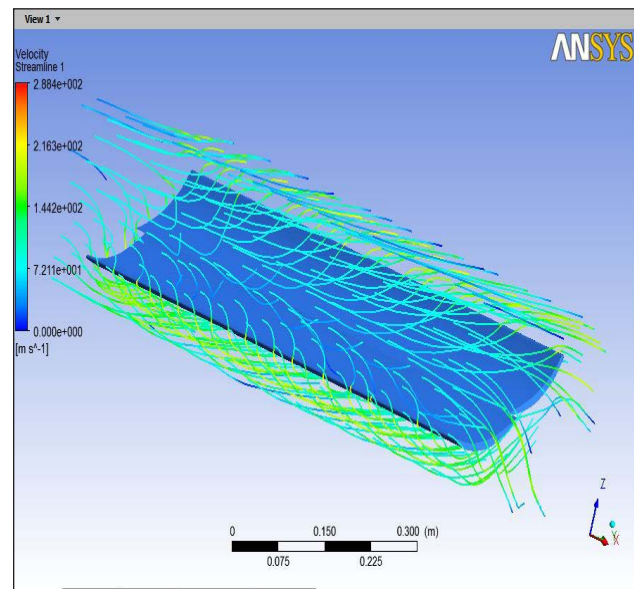


Fig 4. Streamline View for Speed 77.77 m/s with DRS off.

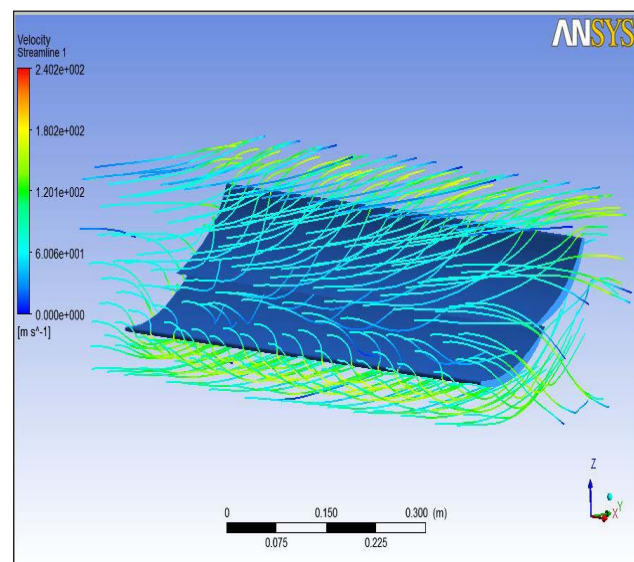


Fig 5. Streamline View for Speed 83.33 m/s with DRS off.

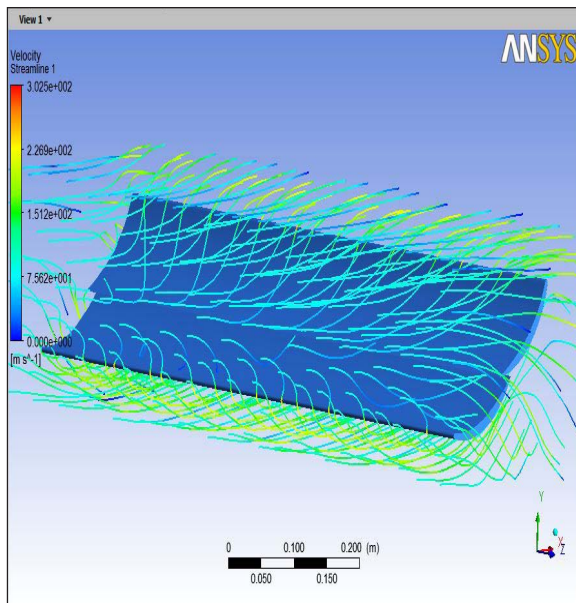


Fig 6. Streamline View for Speed 88.88 m/s with DRS off.

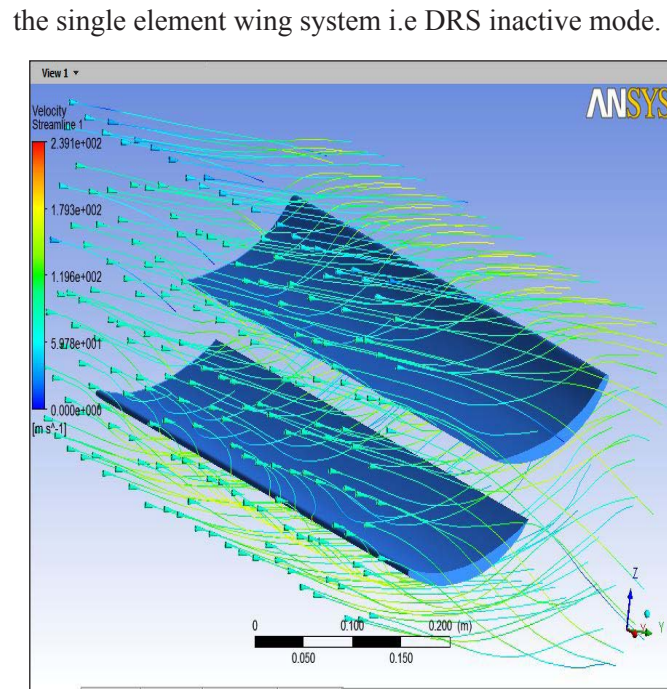


Fig 8. Streamline View for Speed 83.33 m/s with DRS on.

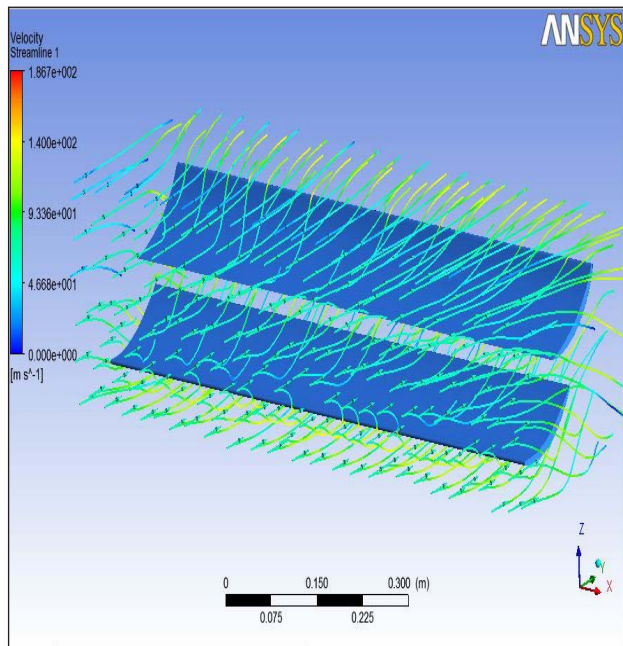


Fig 7. Streamline View for Speed 77.77 m/s with DRS on.

Over the wing span, a gap of 50 mm is pocketed so as to make the single element wing into two element air foil. The lower wing turns into a angle of 15° leaving a space of 50 mm which makes the DRS active. Here in fig 7,8,9 it is shown that the with increase in velocity the streamline gradient also increase but in a lesser rate as compared to

the single element wing system i.e DRS inactive mode. In solver mode the above graphs are obtained. Fig 10, 11, 12 shows the axis velocity when the wings are 10 mm separated. By performing 20 numbers of iterations it is made clear that slight fluctuation is even studied with increase in vehicular velocity. The above said results are obtained when the wings are studied under the fluent analysis or Computational fluid direction.

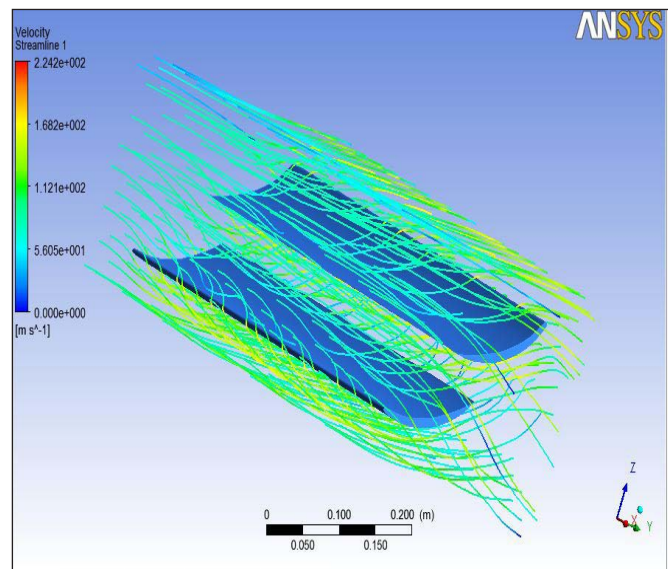


Fig 9. Streamline View for Speed 88.88 m/s with DRS on.

ANSYS GRAPHICAL RESULT WITH 20 ITERATIONS

Though the drag fluctuation caused in velocity lines is very hard to visible, this small drag reduction with the use of DRS matters a lot in increasing the stability and performance of cars.

Here fig 13,14 and 15 shows the velocity lines when the DRS is active. Here also the fluctuation is very hard to visible but yet its effectiveness cause a 16 % reduction in overall air resistance.

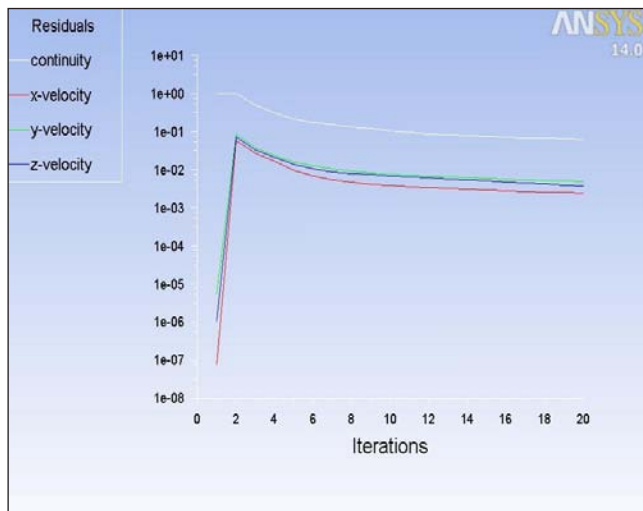


Fig 10. Graphical Result DRS off with 77.77m/s

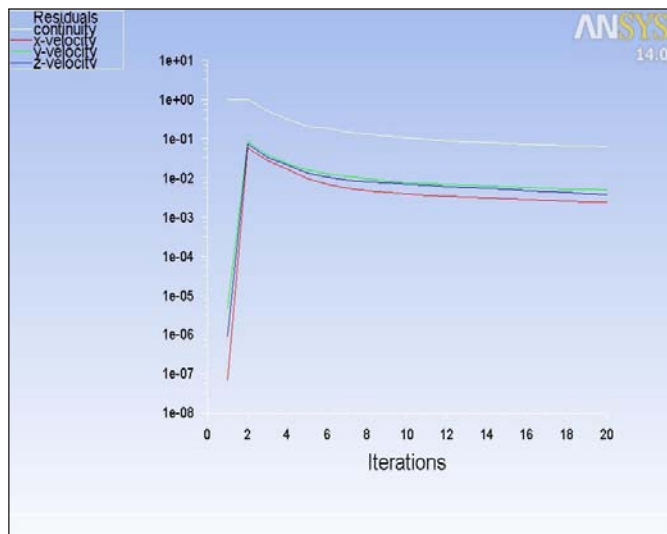


Fig 11. Graphical Result DRS off with 83.33 m/s

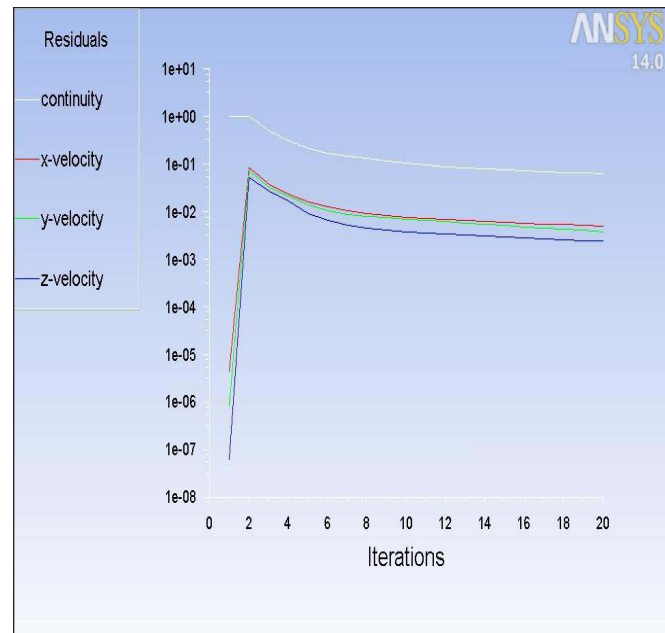


Fig 12. Graphical Result DRS off with 88.88 m/s

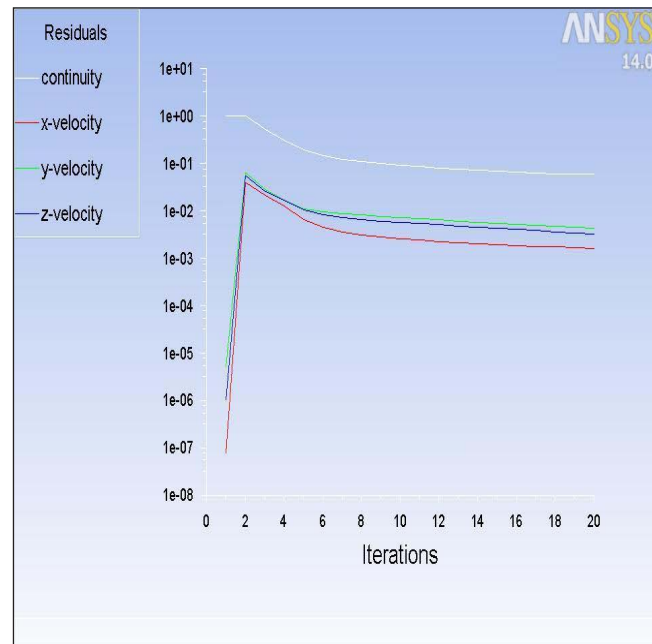


Fig 13. Graphical Result DRS on with 77.77 m/s

With the application of DRS the car rear wing acts somewhat similar to air craft wings. Just the car give a downward force rather than upward force. With proper application of DRS, high level of grip, good traction and better contact with road surface is well maintained.

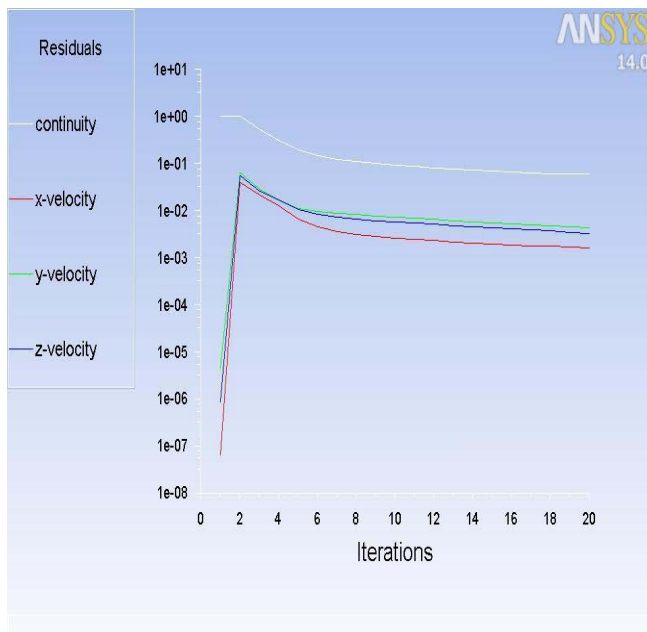


Fig 14. Graphical Result DRS on with 83.33 m/s

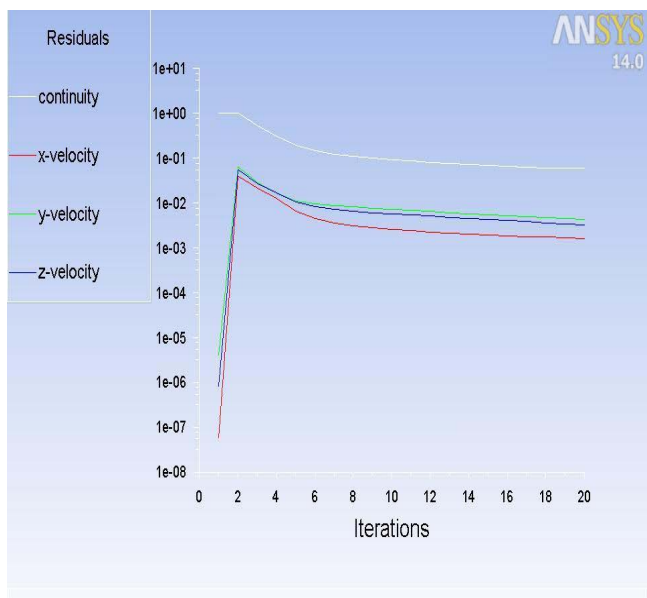


Fig 15. Graphical Result DRS on with 88.88 m/s

CONCLUSION

It is concluded that air resistance is the major concern in increasing drag. By keeping the aerodynamic drag constant ($k=0.393$), it is found that 26 % of air resistance is

decreased by applying DRS. This indicates that installing a DRS in sport and sedan cars are equally important as maintaining a streamline shape for cars. By using DRS, the speed, acceleration, fuel efficiency and stability are increased to a visible margin. More air resistance can be reduced by modifying the wing system which paves the way of future improvement for future researchers.

REFERENCE

- Katz, J. (1995). *Race Car Aerodynamics- Designing for Speed* (1st ed.). Bently Publishers.
- Barbut, D., Negrus, E. M. (2011). CFD analysis for road vehicles - Case study. *Incas Bulletin*, 3(3) 15-22.
- Rouméas, M., Gilliéron, P., & Kourta, A. (2009). Drag reduction by flow separation control on a car after body. *International Journal for Numerical Methods in Fluids*, 60(11), 1222-1240.
- Guilmineau, E. (2008). Computational study of flow around a simplified car body. *Journal of Wind Engineering and Industrial Aerodynamics*, 96(6-7), 1207-1217.
- MunirIslam, M., & Mamun, M. (2010). *Computational Drag Analysis over a Car Body*. International Conference on Marine Technology, Dhaka, Bangladesh, 2010.
- Koike, M., Nagayoshi, T., & Hamamoto, N. (2004). Research on Aerodynamic Drag Reduction by Vortex Generator. *Mitsubishi Motors Technical Reviews*, 16(2004) 13-18.
- Krishnani, P. N. (2009). CFD study of drag reduction of a generic sport utility vehicle, *Thesis for M.Sc. Mechanical Engineering, California State University, Sacramento*.
- Damjanović, D., Kozak, D., Živić, M., ŽeljkoIvandić., Baškarić, T. (2011). *CFD analysis of concept car in order to improve aerodynamics*. JärmüipariInnováció, University of Osijek, Croatia,
- Islam, T., Rakibul Hassan, S. M., & Ali, M. (2012). *Aerodynamic drag of racing cars, global engineering, science and technology conference*, Dhaka, Bangladesh.