DESIGN AND ANALYSIS OF VORTEX GENERATORS FOR REDUCING DRAG FORCE IN AUTOMOBILES BY USING CFD

[1]P. Srinivasa Rao, [2]B. Bharath Kumar

[1]Assistant Professor, Pragati Engineering College (Autonomous), Surampalem, Peddapuram, AP
[2]Assistant Professor, Mahaveer Institute of Science & Technology, Vysapuri, Hyderabad, TS

***Abstract*: In the modern day design of vehicles, especially in the automobile industry involves a great deal of aerodynamic design study to analyze the airflow. The aerodynamic drag force adversely affects the forward motion of the vehicle, which in turn reduces the efficiency. If the vehicle is redesigned to optimize the aerodynamic forces, it could produce better results but requires a huge capital to change the complete design. Here in this paper, we are going to use these vortex generators for the sedan modeled light weighed compact cars with various profiled designs and CFD.**

***Keywords*: Vortex generator, Drag force, airfoil, CFD**

INTRODUCTION

A vortex generator is a device which is used to control the aerodynamic for the vehicles; it is present on the top surface of the vehicle. Generally these are been used in aerodynamic vehicles such as aircrafts and for cars. When the airfoil is in motion relative with the air, the vortex generator creates an vortex, which by removing the part of the slow moving boundary layer in contact with the airfoil surfaces delays local flow separation and aerodynamic stalling, thereby improving the effectiveness of the wings as flaps, elevators and rudders.

Vortex generators are mostly used to delay the flow of air separation which is travelling on the surface of the object. These are used on the external surfaces of the vehicle. These are commonly in rectangular or triangular in shape. These will be placed obliquely, so that they can acquire the angle of attack with respect to the air flowed on the vortex which creates an energy drawing on the tip moving outside in to the boundary layer in contact with the surface.

The study of air travel above the surface of a solid is called aerodynamics. When an automotive moves in a definite velocity the air flow over the car makes drag which is very undesirable for its performance. An automotive needs more power to overwhelm this drag force. When the aerodynamic stuff of the automotive is equipped to overcome this air resistance, the vehicle can move faster, longer and could be added fuel efficient for the vehicle. The vehicle could advance more down force thus providing better grip between the car and the road. The down force allowsthe vehicle to corner at high speeds. However here exists a balance for high speed because of the improved resistance. The aerodynamic stuff of the automotive can be altered by installing a vortex generator at the rear of a car.

Though the main focuses of vehicle manufacturers, many researchers have been focused on fuel saving strategies of the commercial and non-commercial vehicles till to date. As the numbers of passenger cars are being increased considerably in worldwide, it became an important to study the aerodynamic effects of vehicles. Henceforth in this work, the difference of pressure coefficient with respect to the dynamic pressure with different types of vortex generators (VG) on the roof of a sedan vehicle has been investigated.

EXPERIMENTAL DETAILS

Design of vortex generator

In order to discovery a viable configuration, one must first recognize the significant variables for vortex generator design. In order to decrease the degrees of freedom, most of the variables were stationary based on both analysis and references of previous researchers. A Single vane type delta (triangular) shaped was chosen. Due to their uncomplicatedness and widespread usage, the low drag device than any other type makes the vane type more suitable for attributing on the vehicle body. Delta shaped vortex were most usually used in aircraft wings. In linking with the height, the thickness of the limits were measured based on the assumption that the optimum height of the vortex would be almost near to the boundary layer thickness. Below Figure shows the velocity profile on the vehicle's roof. From Figure, the boundary layer thickness at the roof end directly in front of the separation point is found to be about 2mm. Consequently, the optimum height for the VG is estimated to be up to approximately 5mm. The thickness of VG was fixed at 2mm uniform throughout so as to make a stiffened structure.

CFD Analysis of the model using the necessary boundary conditions

CFD is a simulation of fluid engineering system which runs with a mathematical physical problem formulation and numeric methods such as solvers, numerical parameters, grids, etc., Basically we fluid oriented problems will be solved in the fluid analysis. Before that we need to know the physical properties of the fluid which we are going to use in our project. CFD has a lot of advantages are it has been using in the industries like aerospace, automotive, biomedicine, chemical processing, heat ventilation, HVAC, air conditioning systems, hydraulics, marine, etc.,.

In CFD the fluid used will be a liquid or gasses only. Here for these liquids we require the properties like velocity, pressure, temperatures, density, and viscosity.

IMPORTANCE OF CFD

The below table shows the advantages of the CFD

	SIMULATION (CFD)	EXPERIMENTAL
COST	Cheap	Expensive
TIME	Short	Long
SCALE	Any	Small/middle
INFORMATION	All	Measured points
REPEATABLE	Yes	Some
SAFETY	Yes	Some dangerous

Table: shows the comparison of the CFD with Experimental

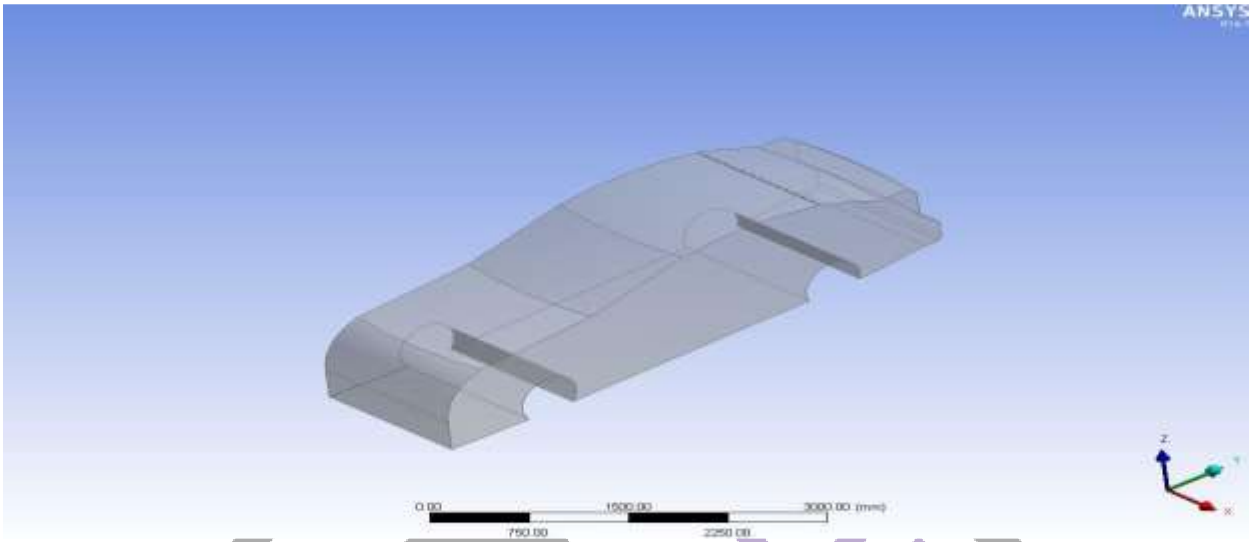


Fig: shows the imported model in to the ansys CFD

MESHING

Meshing is the term automatically integrated with each solver in the ansys. Here there are many times of meshing models. Here in our project we have used a medium based meshing with the triangular profile. The below figures shows the meshing of the imported model and the enclosure.

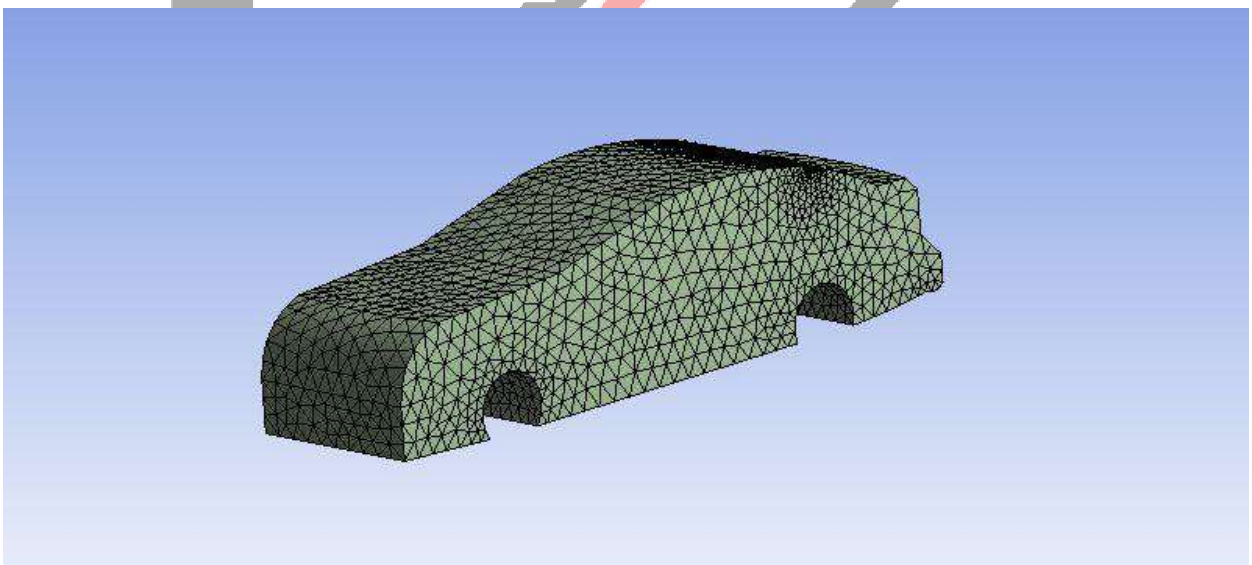


Fig: Shows the meshing of the vehicle

Imported file in to CFD setup

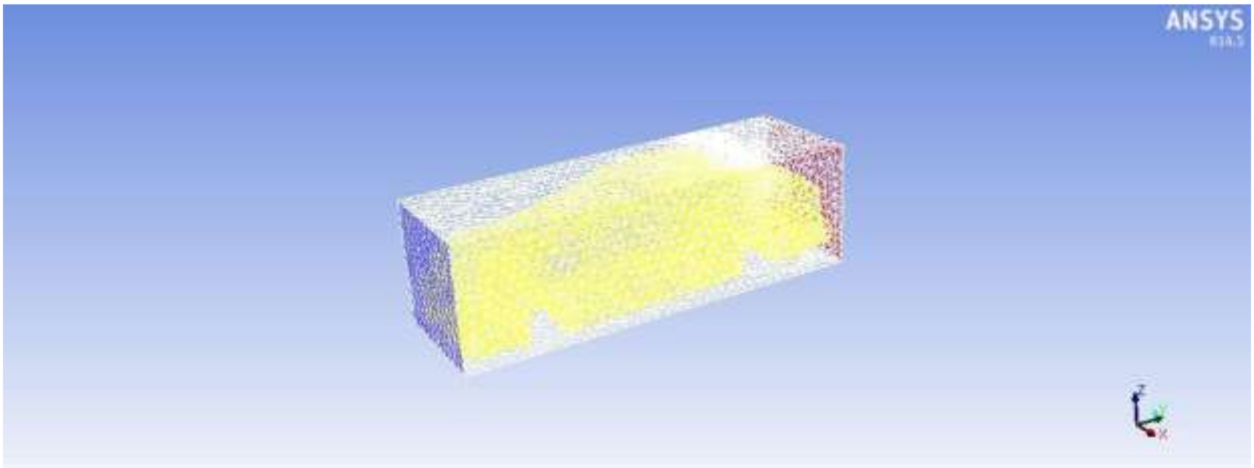


Fig: shows the file which is imported to the setup of fluent

CONVERGENCE GRAPH

Convergence is a universal theory in finite element analysis, as if a model contains any nonlinearity it cannot be solved directly, so here we use to give iterations to get the solution. If we give these inputs we can get the approximate result to the Ansys which we have performed. The convergence criteria explain how close the exact balance is acceptable.

Larger the CC value errors the steps performed, lower the CC value so chances of approximation of the result is higher.

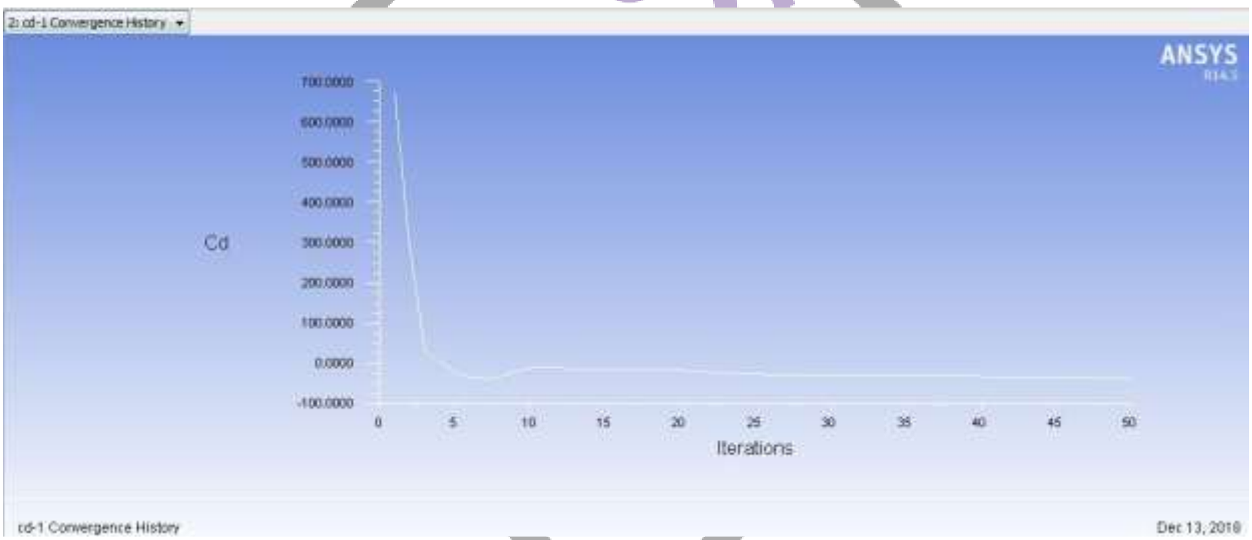


Fig: shows the convergence graph

RESULTS

After all the inputs given to find the result of the Ansys, now we have to consider some outputs which are required to complete our solution and to get the optimum results.

Here we are going to observe the results like

- Pressure
- Stress
- Turbulence kinetic energy
- Velocity
- Drag of the vehicle
- Lift of the vehicle.

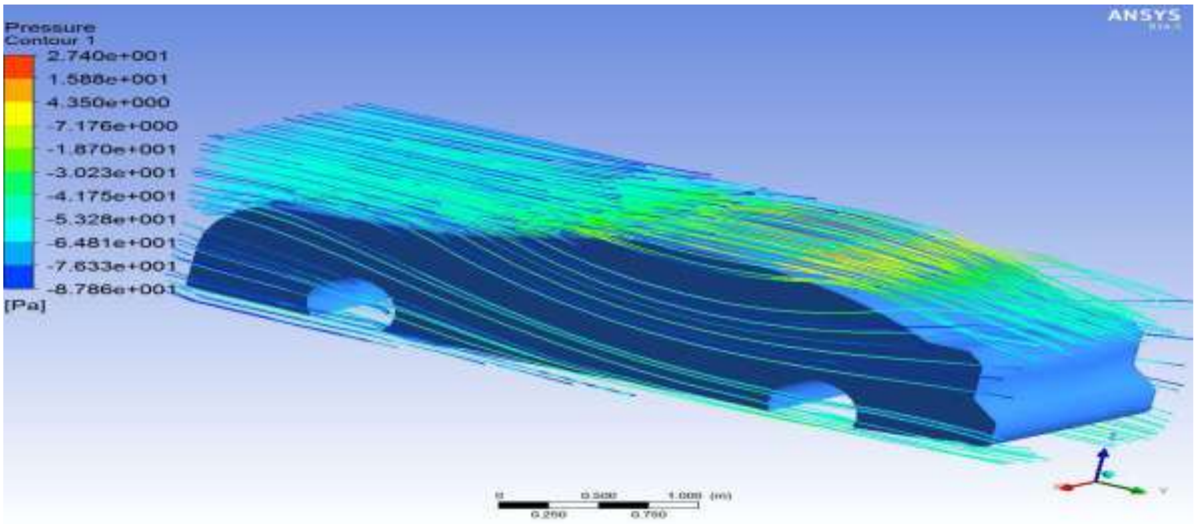


Fig: shows the contours of pressure distribution

ANALYSIS OF THE SECOND MODEL OF VORTEX GENERATOR
CONVERGENCE GRAPH

Convergence is a universal theory in finite element analysis, as if a model contains any nonlinearity it cannot be solved directly, so here we use to give iterations to get the solution. If we give these inputs we can get the approximate result to the ansys which we have performed. The convergence criteria explain how close the exact balance is acceptable.
Larger the CC value errors the steps performed, lower the CC value so chances of approximation of the result is higher.

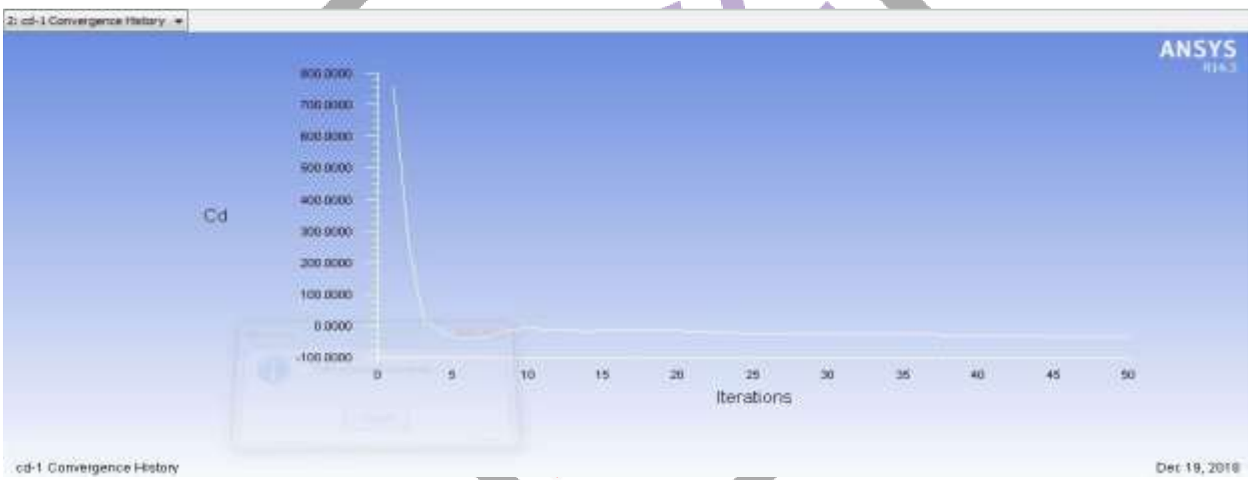


Fig: shows the convergence graph

RESULTS

After all the inputs given to find the result of the ansys, now we have to consider some outputs which are required to complete our solution and to get the optimum results.
Here we are going to observe the results like
- Pressure
- Stress
- Turbulence kinetic energy
- Velocity
- Drag of the vehicle
- Lift of the vehicle

The below figure spots the result, counters of the pressure distribution from minimum to
Maximum As if we see the contours here the maximum pressure is spotted at the roof of the vehicle and even at the placement of the vortex generator.

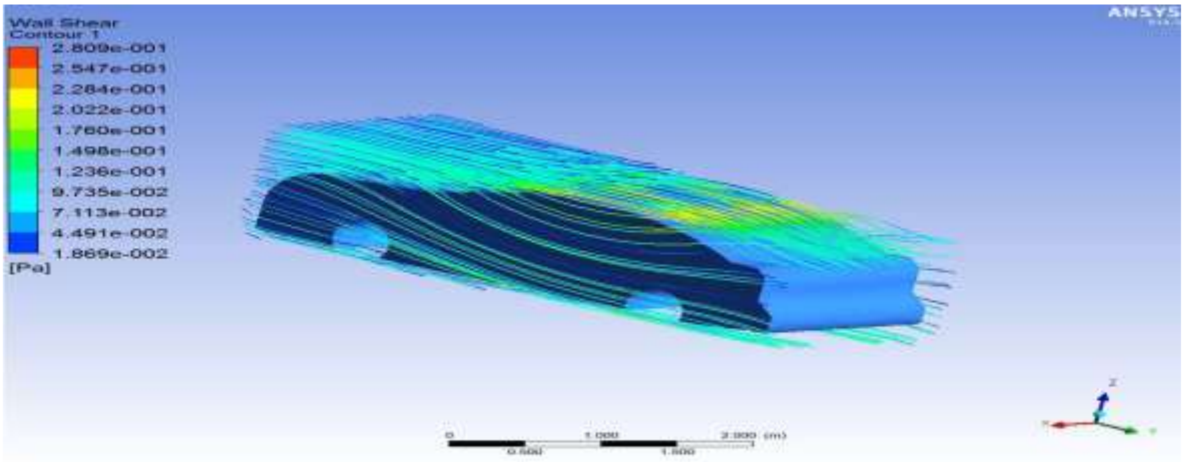


Fig: shows the contours of pressure distribution

The below figure spots the result, counters of the stress distribution from minimum to Maximum. As if we see the contours here the maximum stress is spotted at the roof of the vehicle and even at the placement of the vortex generator.

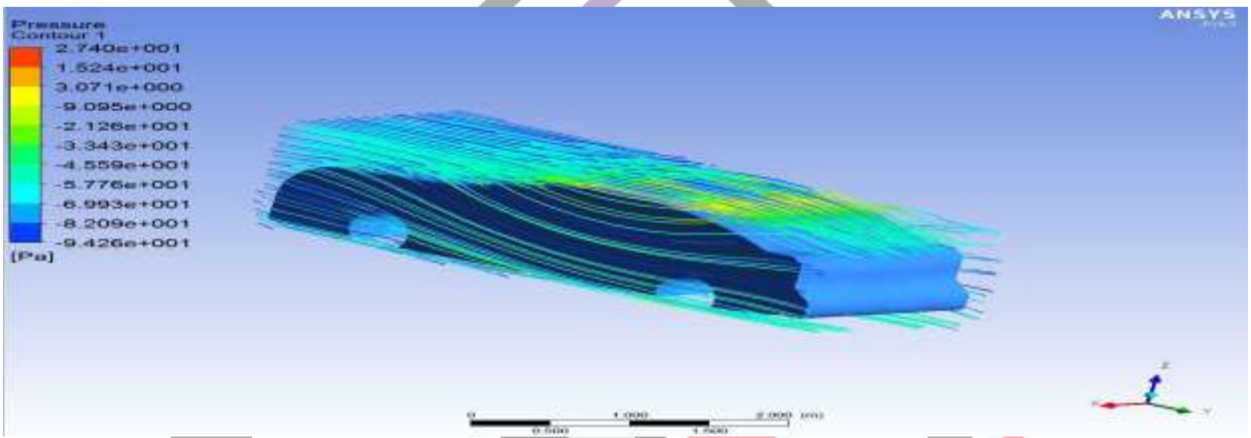


Fig: shows the stress distribution

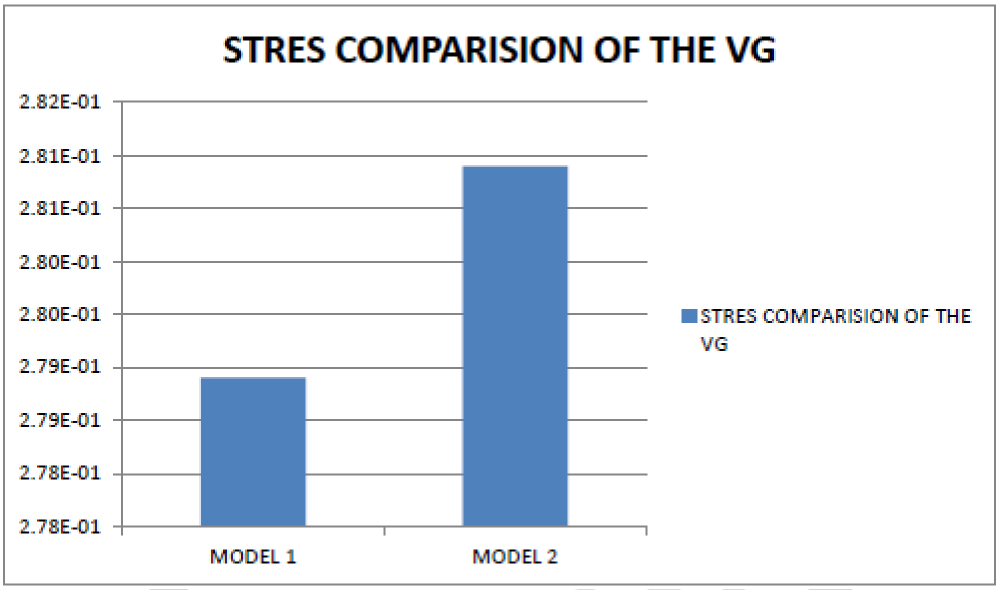
DRAG &LIFT MONITORS

"DRAG - X"
Forces
Forces (n)
Coefficients
Zone Pressure Viscous Total
Pressure Viscous Total
wall-7-shadow (0.45055029 47.077377 38.014282) (0.0056254477 2.3353767 -0.10847144) (0.45617574 49.412754 37.905811) (0.73559231 76.861024 62.064134) (0.0091844044 3.81286 -0.17709623) (0.74477671 80.673884 61.887038)
wall-7 (0 0 0) (0 0 0) (0 0 0)
(0 0 0) (0 0 0) (0 0 0)
wall-13 (4.1214531e-05 0.0045255115 -0.018949121) (-5.5944558e-05 -8.8450193e-05 -0.00012961554) (-1.4730027e-05 0.0044370613 -0.019078737) (6.728903e-05 0.0073885902 -0.030937341) (-9.1338054e-05 -0.00014440848 -0.00021161721) (-2.4049024e-05 0.0072441818 -0.031148958)
wall-12 (0 0 0) (0 0 0) (0 0 0)
(0 0 0) (0 0 0) (0 0 0)
wall-solid (-0.41114667 0 -38.585205) (-2.7425584e-05 3.3717473 -0.10278235) (-0.4111741 3.3717473 -38.687987) (-0.67125987 0 -62.996253) (-4.4776463e-05 5.5048935 -0.16780792) (-0.67130465 5.5048935 -63.164061)
---- --- --- ---------------- ---------------------------- --

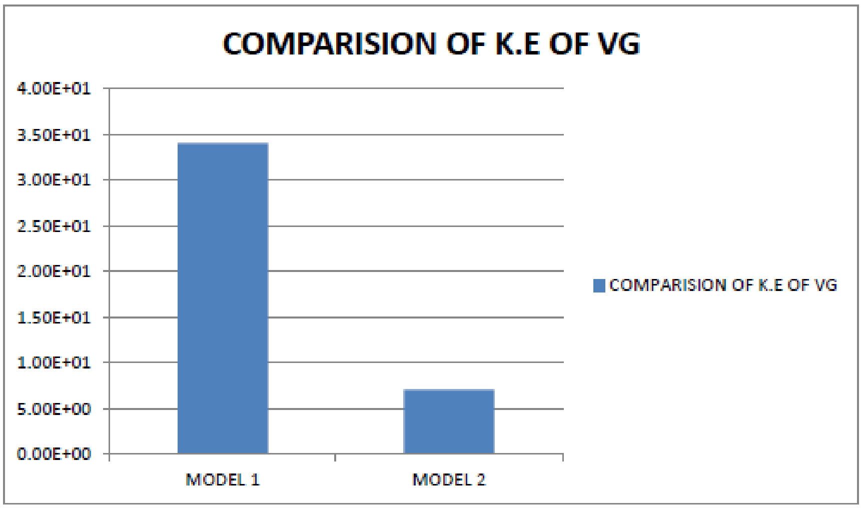
Net (0.039444832 47.081903 -0.58987197) (0.0055420776 5.7070355 -0.21138341) (0.044986909 52.788938 -0.80125538) (0.064399726 76.868413 -0.96305628) (0.0090482899 9.3176091 -0.34511577) (0.073448016 86.186022 -1.308172)

4.1 GRAPHICAL COMPARISON

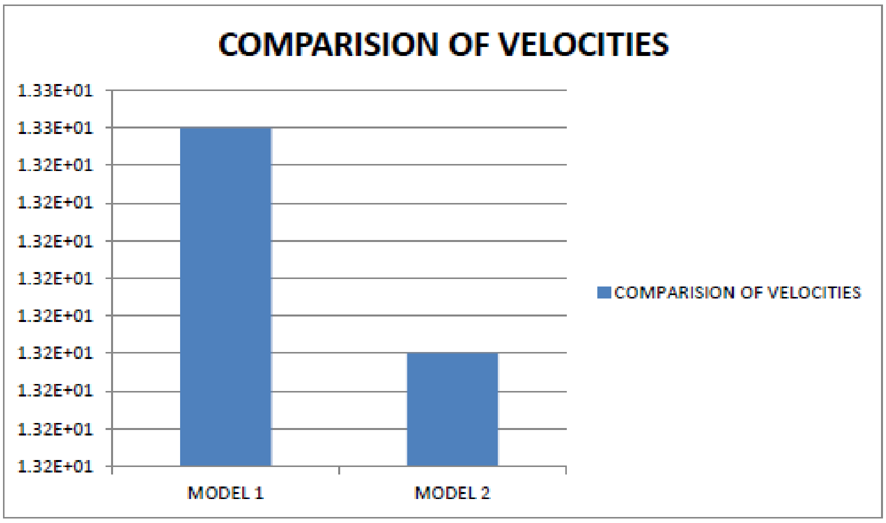
4.1.1 STRESS COMPARISION



4.1.2 COMPARISION OF TURBULENCE KINETIC ENERGY



4.1.3 COMPARISION OF VELOCITIES



4.2 CONCLUSION

Here in this thesis we are going to use these vortex generators for the sedan modeled light weighed compact cars with various profiled designs and CFD will be carried out for the prediction.

As if we verify the project analysis results in the graphical format, here the pressure on the front bodies is same in both the models and there is not any change, as if we verify the stress and the velocities on the vehicle, there we can clearly observe that the air should flow with the higher velocities and the stress on the vehicle should be low and when we compare the velocities in the graphical format, here we can conclude that the model 1 has the higher velocity values than the model 2 and even in terms of pressure and stress the model 1 satisfies the boundary conditions, so here we can conclude that according to the drag and lift of the vehicle forces also model 2 is the suggestible for the better safety and free aerodynamic flow of air.

REFERENCES AND CITATIONS

1. Taylor H D 1947 The elimination of diffuser separation by vortex generators United Aircraft Corp. Report No.R-4012-3
2. Oye S 1995 the effect of vortex generators on the performance of the ELKRAFT 1000 kW turbine 9th IEA Symp. Aerodynamics of Wind Turbines (Stockholm) ISSN0590- 8809
3. Lin J C 2002 Review of research on low-profile vortex generators to control boundary-layer separation Prog. Aerosp. Sci. 38 389–420
4. Godard G and Stanislas M 2006 Control of a decelerating boundary layer. Part 1: optimization of passive vortex generators Aerosp. Sci. Technol. 10 181–91
5. Angele K P and Grewe F 2007 Instantaneous behavior of streamwise vortices for turbulent boundary layer separation control J. Fluids Eng. 129 226–35
6. Schmidt J J 1997 Experimental and numerical investigation of separated flows PhD Thesis Department of Mechanical Engineering ET-PHD 97-01, MEK-DTU
7. Raffel M, Willert C and Kompenhans J 1998 Particle Image Velocimetry—A Practical Guide 3rd printing (Berlin:Springer)
8. Keane R D and Adrian R J 1990 Optimization of particle image velocimeters. Part I: double pulsed systems Meas. Sci.Technol. 1 1202–15
9. Westerweel J 2000 Theoretical analysis of the measurement precision in particle image velocimetry Exp. Fluids29 S3–12.
10. Foucaut J, Miliat B, Perenne N and Stanislas M 2004 Characterization of different PIV algorithms using the EUROPIV synthetic image generator and real images from a turbulent boundary layer Particle ImageVelocimetry: Recent Improvement. Proc.EUROPIV 2Workshop on Particle Image Velocimetry (Zaragoza,March–April 2003) ed M Stanislas et al (Berlin: Springer)
11. Stanislas M, Okamoto K and Khler C 2003 Main results of the first Int. PIV challenge Meas. Sci. Technol. 14 R63–89
12. Durst F and Schafer M 1996 A parallel block-structured multigrid method for the prediction of incompressible flows Int. J. Numer. Methods Fluids 22 549–65
13. Lund T S, XiaohuaW and Squires K D 1998 Generation of turbulent inflow data for spatially-developing boundary layer simulations J. Comput, Phys. 140 233–58
14. Cavar D 2006 Large eddy simulation of industrially relevant flows PhD Thesis Department of Mechanical Engineering MEK-FM-PHD 2006-02, MEK-DTU
15. Smagorinsky J 1963 General circulation experiments with the primitive equations Mon. Weather Rev. 91 99–165
16. Gad-el-HakM and Bandyopadhyay P R 1994 Reynolds number effects in wallbounded turbulent flows Appl. Mech. Rev. 47 307–65.
17. M. Knepper, Examination of Three Candidate Technologies for High-Lift Devices on an Aircraft Wing. Cranfield University (2005)
18. P. T. Soderman, Aerodynamic Effects of Leading-Edge Serrations on a Two- Dimensional Airfoil. NASA (1972).
19. S. Brennan. A wind tunnel investigation into the effect of a serrated trailing. MSc Thesis. Cranfield University (2002).

20. B. J. Hoskins, M. E. Mcintyre, A. W. Robertson. RMS, On the use and significance of isentropic potential vorticity maps. 111, 470 (1985)
21. J. D. Anderson. Introduction to Flight. 368-370 (2008).
22. Boeing. AERO, Exceeding Tire Speed Rating During Takeoff. (2009).