© May 2019 | IJIRT | Volume 5 Issue 12 | ISSN: 2349-6002

Analysis of Reduction in Drag and Improvement in Vehicle Stability Using Vortex Generator Vehicle

Bhushan M. Patil[1], A. V. Patil[2]
[1] *M. Tech, Design Engineering, S. S. G. B. C. O E & T, Bhusawal, Maharashtra, India*
[2] *Former Professor, Department of Mechanical Engineering, S. S. G. B. C. O E & T, Bhusawal, Maharashtra, India*

***Abstract*- The main reason for the aerodynamic drag in the car is that the flow separation on the rear of the vehicle body. In order to delay the disruption of this flow, the Vortex Generator is used in recent vehicles. Vortex generators are commonly used in aircraft to prevent the separation of the flow. vortex generators create their own drag, but due to splitting the stream on the downstream, the drag also reduces. The effect on the shape, size, and orientation of the vortex generator is dependent on. CFD analysis confirms that the use of vertex generators reduces both the drag & lift coefficient. Therefore, an optimized size is required with the right orientation to get great results. This paper represents the effect of the flow of generators on a different orientation and the effect which has the effect.**

Index Terms- Aero dynamics, body, Computational Fluid Dynamics (CFD), Flow simulation.

I. INTRODUCTION

A vertex generator (VG) is an aerodynamic device consisting of a small vanes usually attached to lifting surface can airfoil, such as an aircraft wings or a rotor blade of wind turbine.

VG may also be attached to some part of an aerodynamic vehicle such as an aircraft & use it at ledge or a car. When the airfoil or the body is in motion relative to the air, the VG creates a vertex, which is by removing some part of slow moving Boundary layer is in contact with airfoil shape surface delay local flow separation & aerodynamic stalling during effectiveness of wings and control surfaces such as flaps, elevators, aitrerons & rubbers.

CFD: Computational fluid dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and data structures to analyze and solve problems that involve fluid flow.

CFD ANALYSIS PROCESS

The general process for performing a CFD analysis is outlined below so as to provide a reference for understanding the various aspects of a CFD simulation. The process includes: Formulate the Flow Problem.

A. Model the Geometry and Flow Domain
B. Establish the Boundary and Initial Conditions
C. Generate the Grid
D. Establish the Simulation Strategy
E. Establish the Input Parameters and Files
F. Perform the Simulation
G. Monitor the Simulation for Completion
H. Post-process the Simulation to get the Results
I. Make Comparisons of the Results
J. Repeat the Process to Examine Sensitivities
K. Document

Formulate the Flow Problem

The first step of the analysis process is to formulate the flow problem by seeking answers to the following questions:

- What is the objective of the analysis?
- What is the easiest way to obtain those objectives?
- What geometry should be included?
- What are the free stream and/or operating conditions?
- What dimensionality of the spatial model is required? (1D, quasi-1D, 2D, axisymmetric, 3D)
- What should the flow domain look like?
- What temporal modeling is appropriate? (steady or unsteady)
- What is the nature of the viscous flow? (in viscid, laminar, turbulent)

Model the Geometry and Flow Domain

The body about which flow is to be analyzed requires modeling. This generally involves modeling the geometry with a CAD software Shown in fig.No.1. Approximations of the geometry and simplifications may be required to allow an analysis with reasonable effort. Concurrently, decisions are made as to the extent of the finite flow domain in which the flow is to be simulated. Portions of the boundary of the flow domain coincide with the surfaces of the body geometry. Other surfaces are free boundaries over which flow enters or leaves. The geometry and flow domain are modeled in such a manner as to provide input for the grid generation. Thus, the modeling often takes into account the structure and topology of the grid generation.

Bounding Box length X, Y, Z axis 0.3m, 0.16m, 0.149m respectively.

Statistics: Bodies no. 2, Active Bodies 2 nos, and nodes 240073nos. , Elements 1320338 nos.

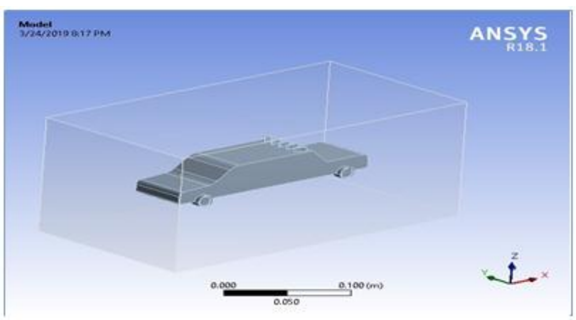


Fig.1 Model of Car Bodies

Establish the Boundary and Initial Conditions

Since a finite flow domain is specified, physical conditions are required on the boundaries of the flow domain. The simulation generally starts from an initial solution and uses an iterative method to reach a final flow field solution.

Generate the Grid

The flow domain is discretised into a grid. The grid generation involves defining the structure and topology and then generating a grid on that topology. Currently all cases involve multi-block, structured grids; however, the grid blocks may be abutting, contiguous, non-contiguous, and overlapping. The grid should exhibit some minimal grid quality as defined by measures of orthogonality (especially at the boundaries), relative grid spacing (15% to 20% stretching is considered a maximum value), grid skewness, etc... Further the maximum spacing's should be consistent with the desired resolution of important features. The resolution of boundary layers requires the grid to be clustered in the direction normal to the surface with the spacing of the first grid point off the wall to be well within the laminar sublayer of the boundary layer. For turbulent flows, the first point off the wall should exhibit a y+ value of less than 1.0.

Establish the Simulation Strategy

The strategy for performing the simulation involves determining such things as the use of space-marching or time-marching, the choice of turbulence or chemistry model, and the choice of algorithms.

Establish the Input Parameters and Files

A CFD codes generally requires that an input data file be created listing the values of the input parameters consisted with the desired strategy. Further the grid file containing the grid and boundary condition information is generally required. The files for the grid and initial flow solution need to be generated.

Perform the Simulation

The simulation is performed with various possible with options for interactive or batch processing and distributed processing.

Monitor the Simulation for Completion

As the simulation proceeds, the solution is monitored to determine if a "converged" solution has been obtained, which iterative convergence. Further discussion can be found on the page entitled Examining Iterative Convergence.

Post-Process the Simulation to get the Results

Post-Processing involves extracting the desired flow properties (thrust, lift, drag, etc...) from the computed flow field.

Make Comparisons of the Results

The computed flow properties are then compared to results from analytic, computational, or experimental studies to establish the validity of the computed results.

Repeat the Process to Examine Sensitivities

The sensitivity of the computed results should be examined to understand the possible differences in the accuracy of results and / or performance of the computation with respect to such things as: dimensionality , flow conditions , initial conditions , marching strategy , algorithms , grid topology and density , turbulence model , chemistry model , flux model , artificial viscosity , boundary conditions , computer system.

STEPS FOLLOWED IN ANALYSIS

a. Creating Geometry
1. Geometry draws in catia.
2. Select X-Y plan for draw geometry,
3. Dimension

b. Mesh
1. Go in model option
2. In Mesh Control, Select Refinement
3. In refinement Select surface body and click apply in Geometry, give refinement value 2.

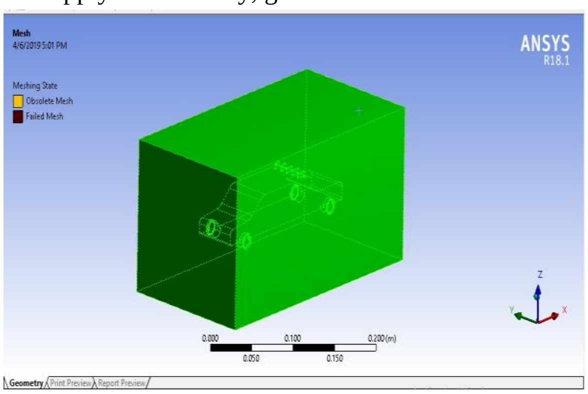


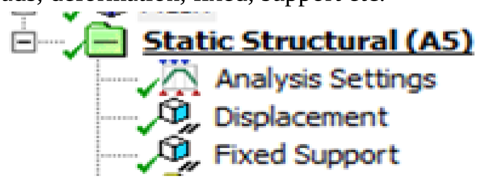
Fig . 2 Meshing of Car Bodies

b.1 Details Meshing

- Size Function Curvature
- Relevance Center Coarse
- Transition Slow
- Span Angle Center Fine
- Curvature Normal Angle Default (18.0 °)
- Min Size Default (1.8503e-004 m)
- Max Face Size Default (1.8503e-002 m)
- Max Tet Size Default (3.7006e-002 m)
- Growth Rate Default (1.20)
- Automatic Mesh Based Defeaturing On
- Defeature Size Default (9.2514e-005 m)
- Minimum Edge Length 6.589e-005 m
- Quality
- Check Mesh Quality Yes, Errors
- Target Skewness Default (0.900000)

c. Static structural

In the static structural condition position of the geometry & there displacement can be decided the loads, deformation, fixed, support etc.

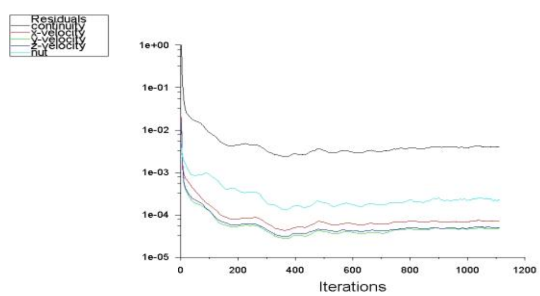


III. COORDINATE SYSTEM

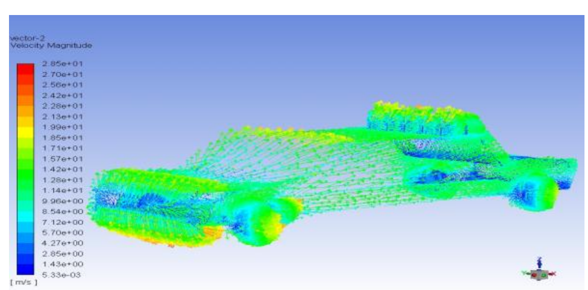
Table 1.0 Coordination System

Object Name	Global Coordinate System
State	Fully Defined
Definition	
Type	Cartesian
Coordinate System ID	0.
Origin	
Origin X	0. m
Origin Y	0. m
Origin Z	0. m
Directional Vectors	
X Axis Data	[1. 0. 0.]
Y Axis Data	[0. 1. 0.]
Z Axis Data	[0. 0. 1.]

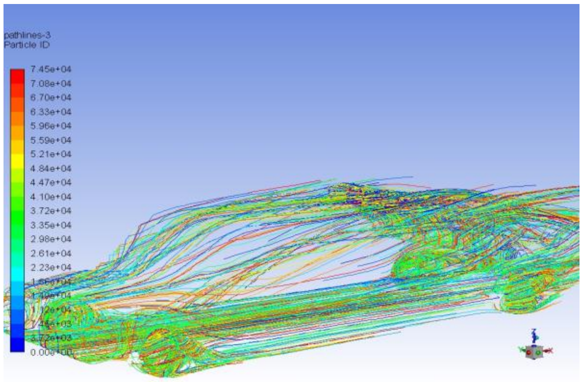
IV. ITERATION FLOW



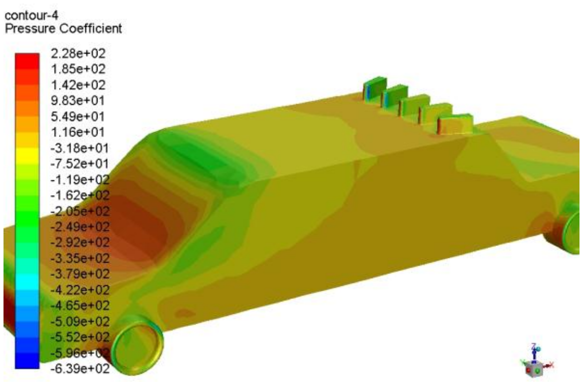
V.RESULTS & CONCLUSION



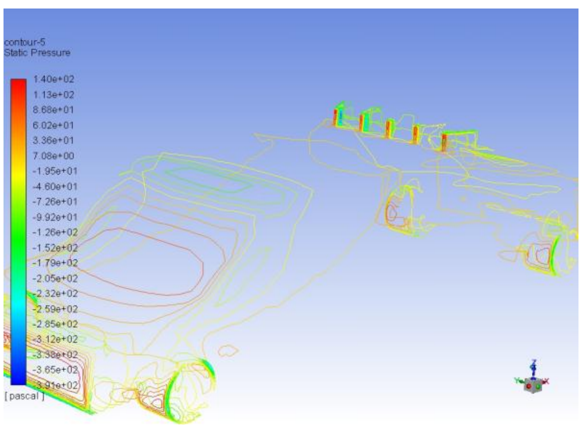
Vector-2 velocity Magnitude



Pathlines-3 Particle ID



Contour-4 Pressure Coefficient



Contour-5 Static Pressure

The velocity distribution plot show that the velocity has increased at the rear end of the vehicle thereby reducing the drag drastically. The drag force obtained for this simulation with the vortex generators. The reduction in drag greatly reduces the air resistance on the car and results in higher top speed and power required in greatly reduced. This also has a direct impact in increased fuel economy

Object Name	Enclosure\Enclosure	FFF\PartBody/EdgeFillet.37
State	Meshed	Fully Defined
	Graphics Properties	
Visible	Yes	
Transparency	0.1	1
Definition		
Suppressed	No	
Coordinate System	Default Coordinate System	
Behavior	None	
Reference Frame	Lagrangian	
Material		
Fluid/Solid	Defined By Geometry (Fluid)	Defined By Geometry (Solid)
Bounding Box		
Length X	0.3 m	0.2 m
Length Y	0.16 m	6.0001e-002 m
Length Z	0.149 m	4.9002e-002 m
Properties		
Volume	6.8763e-003 m³	2.7575e-004 m³
Centroid X	3.4224e-006 m	-8.5284e-005 m
Centroid Y	-2.2406e-008 m	5.3107e-007 m
Centroid Z	2.0875e-003 m	-1.6212e-004 m
Statistics		
Nodes	240073	0
Elements	1320338	0
Mesh Metric	None	
CAD Attributes		
Part Tolerance:	0.00000001	

VI. CONCLUSION

From the simulations performed we can conclude that the designed vortex generators have satisfied their purpose by reducing the drag, increasing the downforce and reducing the boundary layer separation. By observing the pressure over and under the car the coefficient of pressure can be found for both the models with the vortex generators.

REFERENCE

[1] B.M.Patil, A.V.Patil, "Analysis of Reduction In Drag And Improvement In Vehicle Stability Using Vortex Generator Vehicle", in IJIRSET, Vol.1 Issue, April 2019.

[2] Mohan Jagadeesh Kumar M, Anoop Dubey, Shashank Chheniya, & Amar Jadhav, "Effect of Vortex generators on Aerodynamics of a Car: CFD Analysis", 'International Journal of Innovation in Engineering and Technology.

(IJIET), Vol.2 Issue 1 April 2013, ISSN: 2319-1058.

[3] Raju Govindharajan, Dr.K.M. Parammasivam, S. Sathya Narayanan" Design of Vertex Generators For Light Transport Vehicles (LTVs) using CFD", 'APCWE-VIII', ' The Eighth Asia-Pacific Conference on Wind Engineering, December 10-14,2013, Chennai, India.

[4] Hardik Panchal, Krishna Kumar & Raybahadursinh chauhan, "A Review on Aerodynamic Study of Vehicle Body using CFD", 'NCEVT'14'author profiles for this publication at: 'https://www.researchgate.net/publication/291337928' uploaded by Krishna Kumar on 21 January 2016.