OPTIMAL CONFIGURATION INVESTIGATION OVER GENERIC SIMPLIFIED AHMED BODY

NITISH V KARAMBALI¹, AKSHAY K SAVEKAR², SARTHAK R KHATOD³, PROF. DR. SACHIN
PAWAR⁴

Department of Mechanical Engineering, Mit School of Engineering, Affiliated with MIT-ADT University, India

ABSTRACT

Ahmed Body is generic simplified version of Hatchback vehicle to determine the aerodynamic performance with respect to localized lift and drag creation. The past research carried out on Ahmed body was only focused to determine fluctuating lift and drag creation with respect to angle of attack in slant angle. The present research is preliminary focused to present a scientific co-relation between Lift-Drag-Vortices because the creation of vortices in wake of sheading cycle adversely effect lift and Drag quantities.

KEYWORDS: *Optimal, Body Drains & Generic*

Received: Nov 22, 2020; **Accepted:** Dec 12, 2020; **Published:** Dec 18, 2020; **Paper Id.:** IJMPERDDEC202042

1.INTRODUCTION

Ahmed's body is a standard car body (simplified car model). The air flow around Ahmed's body captures the vital flow elements around the car and was first described and seen in Ahmed's experimental work. Despite its simplicity, Ahmed's body allows us to capture the physical features of the automotive industry. pressed between the anterior and posterior part.

Ahmed's body model was developed to represent cars as a blue body in a drag-and-drop price survey. According to him, most of the drag from Ahmed's body model is done behind the body and almost every pull of the body drains the pressure. Since then, Ahmed's body has served as an effective measure for a variety of powerful motor studies. Ahmed Body's computer simulation was performed at a variety of angles, altitudes, and rear angles.

The authors have suggested using a more sophisticated dynamic model in order to accurately capture the partitions and reunions responsible for dragging the back end of Ahmed Body.

Previous attempts have been made to attempt to replicate the position of the corner in the air tunnel using curved models [1], and curved test strips (Bird et al., 1951), but these methods cannot represent all aspects of true corner flow. Recently there has been a development towards a unique new type of air tunnel that is designed to accurately restore the shape of a corner [2]

In the study of public domain in automotive aerodynamics it tends to favor simple bodies over detailed car geometry. This confirms that conclusions are unlikely to be determined by geometry and that the findings may be generalized aerodynamic markers that are widely used (Good and Garry, 2004). Ahmed's body is one of the most widely studied bodies, first analyzed in [3]. The geometry is designed to maintain the flow attached to the front of

the car and allows for a detailed investigation of the flow signs that occur on the sloping rear surface, which is a common feature in hatchbacks/fastbacks. Since this is the geometry chosen for the current study it is appropriate to introduce the main features of the flow. Basic fluid power and various engineering applications show good interest in controlling the flow of [4][5][6][7]; Many strategies have tested to control the separation of the flow or to prevent it or by reducing its effects. These methods are based on the use of synthetic devices for use of active or weak control devices. Performance devices, due to their complexities have to do a lot change the position of the car and frequency is not important when the flow configuration changes, provide the active control devices more attention. Various effective studies of effective control have been made using Ahmed's body at a 25° slant angle. [9] used a consistent wish over the slide to find out by prices a 17% reduction in drag. Aubrun et al. (2011) activate the microjet configuration below the partition line between the roof and the back window is relaxed. They benefited from the pull a reduction of 9~14%, depending on the number of Reynolds. [8] conducted experimental studies with periodic beating jets at the upper edge of the base reached a 5.7% reduction in drag.

2. METHODS AND NUMERICAL SCHEME

2.1 Problem Description

The present research work is focused to find out most optimum arrangement of slant angle by coupling Linear Regression model which resultant in increase Lift Coefficient in comparative study. The investigation is performed using Commercial coded Ansys Fluent CFD Software.

Figure 1: Geometry creation

2.2 CFD investigation

The CFD investigation is divide in 5 major steps as shown in figure below

Table 1 CFD Steps

2.3 Mesh creation

Figure 2: Mesh Creation

In any CFD Investigation the discretization of whole geometry in finite volume chunks is one of the most important steps which makes sure that traveling of information is achieved correctly through nodal points. In present study a fine tetrahedral mesh creation is done by using additional refinement layer of 3 to the boundary fluid flow region to capture all viscous forces associated in shear layers. The skewness factor below 0.90 has been successfully maintained in geometry.

2.4 Numerical Scheme

The present study is solved using commercial coded Newton Raphson technique in Ansys Fluent 18.0 Software with selective Reynolds Averaging Navier–Stokes (RANS) working model. The solution is derived for turbulent inlet fluid flow with 10% of total intensity of having K-Omega Turbulence Model.

Through specific, by incorporating averaged and fluctuating quantities to produce the Reynolds-averaged Navier–Stokes (RANS) equations, the turbulence model aims to modify the original unsteady Navier–Stokes equations. Only when still modelling turbulence influence, without the need for the resolutions of turbulence fluctuations, do these equations represent the mean flow quantities. Both scales are built for the turbulence sector. The turbulence model based

on the RANS equations is called statistical turbulence models because of the statistical averaging technique used to achieve these equations.

In near-wall treatment, the typical k- Ω model is best used. Other benefits include higher efficiency for dynamic limit layer flows under unfavorable pressure gradients and separations (e.g., external aerodynamics and turbo machinery). Rather this model also shows that unnecessary and early division is expected.

The formulation of Kinetic K- Ω model is showed in below equation.

The turbulent energy k is given by:

$$k = \frac{3}{2}(UI)^2 \quad (1)$$

where U is the mean flow velocity and I is the turbulence intensity.

The turbulence intensity gives the level of turbulence and can be defined as follows:

$$I = u'/U \quad (2)$$

where u' is the root-mean-square of the turbulent velocity fluctuations given as:

$$u' = \sqrt{\frac{1}{3}(u_x'^2 + u_y'^2 + u_z'^2)} = \sqrt{\frac{2}{3}k} \quad (3)$$

The mean velocity U can be calculated as follows:

$$U = \sqrt{U_x^2 + U_y^2 + U_z^2} \quad (4)$$

The specific turbulent dissipation rate can be calculated using the following formula:

$$\omega = C_\mu^{\frac{3}{4}} \frac{k^{\frac{1}{2}}}{l} \quad (5)$$

where, C_μ is the turbulence model constant which usually takes the value 0.09, k is the turbulent energy, l is the turbulent length scale.

The turbulence length scale describes the size of large energy-containing eddies in a turbulent flow.

The turbulent viscosity ν_t is, thus, calculated as:

$$\nu_t = k/\omega \quad (6)$$

2.5 Boundary Condition

The Initial boundary for solution Monitoring is used with Inlet Velocity of 50 M/s with Inlet domain as a velocity Inlet and Outlet domain as a Pressure Outlet. The numerical Scheme is prepared using Coupled flow of second order upwind PDE Implicit scheme.

To ensure of capturing shear layer vortices in fluid flow domain an Omega Kinetic dissipation with time scale factor of 0.005 Second is taken. Below figure shows the initial fluid flow boundary condition with given Inlet and Outlet Faces. The Wall side faces is kept with the domain layer of inner fluid interaction cell zone conditions. The Numerical scheme is solved using on ambient fluid flow condition.

3. RESULTS AND DISCUSSIONS

Configuration under Investigation

Slant Angle	CL	CD	Vortices
20 Degree	7.0180815	0.40321498	12.960795
25 Degree	6.521593	0.59685157	13.423324
30 Degree	5.7016673	0.66945003	14.16277
35 Degree	4.8318652	0.76687231	14.3032
40 Degree	4.5426702	0.96460668	15.425106
45 Degree	3.4054909	1.0776355	15.397452

Configuration 1)

Slant Angle Kept 20 Degree

Figure 3: Pressure Contour Figure 4. Velocity Contour

Configuration 2)

Slant Angle Kept 25 Degree

Figure 5: Pressure Contour Figure 6. Velocity Contour

Configuration 3)

Slant Angle kept 30 degree

Figure 7: Pressure Contour Figure 8. Velocity Contour

Configuration 4)

Slant Angle kept 35 degree

Figure 9: Pressure Contour Figure 10. Velocity Contour

Configuration 5)

Slant angle kept 40 degree

Figure 11: Pressure Contour Figure 12. Velocity Contour

Configuration 6)

Slant angle kept 45 degree

Figure 13: Pressure Contour Figure 14. Velocity Contour

CL and CD with respect to Slant Angle configuration

Creation of Vortices on Coefficient of Drag

- Discussion

The Validation of Results are generated by [10] Wind Tunnel Data.

4. CONCLUSIONS

Aerodynamic is one of the important sources term in automotive industry because the 70 % of resistance on Automotive is provided by air. By increasing Lift coefficient scientist can save large amount of fuel economy. The present study is engrossed to performed Aerodynamic investigation over generic simplified version of Hatchback model which is called Ahmed body. The following points are drawn from present investigation –

- By performing Aerodynamic Investigation the co-relation has been setup with increasing slant angle where it is has been found out the optimum range of slant Angle is from 20 Degree to 25 Degree.
- The Lift Coefficient is tend to increase and maintained in the range of 20 – 25 Degree where the Drag Coefficient become Dominance by increasing Slant Angles.
- The reason behind the dominance of Drag coefficient can be vital by understanding Vector field or vortices generation in Wake region
- The co-relation setup between Drag-Lift and Vortices suggest that by increasing Slant Angles the Wake of Sheading Vortices increase due to which the dominance of Drag has been observed by increasing Slant Angles. This phenomenon of Sheading cycle is Accurately measure by Time Depended feature of 0.005 Seconds, Pressure outlet Couple Scheme.

REFERENCES

1. Ahmed, S.R, & Faitin, G. (1984). *Some salient features of the time-averaged ground vahiicle wake*. SAE Technical Paper, 840300.
2. Collis, S. J. (2004). *Issues in active flow control*:. 237-289.
3. Gad-El-Hak, M. (1996). *Modern developments in flow control*. Appl. Mech. Rev. 367-379.
4. Glezer, A. A. (2003). *Synthetic jets*. Annu. Rev. Fluid Mech. 259-289.
5. Gordes, A. (2005). *Process for simulating curved airflow on wheeled in fluid channels with a straight measuring section*.
6. Keogh, James, Barber, T., Diasinos, S., & Doig, G. (2016). *A new type of wind tunnel for evaluation of curved motion* . 54th AIAA Aerospace Sciences Meeting , 1774.
7. Krentel, D. M. (2010). *Application of active flow control on generic 3D car models*. Active Flow Control II. Notes on Numerical Fluid Mechanics and Multidisciplinary Design 108 Springer,.
8. P. Cathalifaud, P. L. (n.d.). *Optimal control by blowing and suction at the wall of algebraically growing boundary layer distribution*. International Union of Theoretical and Applied Mechanics). Springer, Berlin, Heidelberg.
9. Roumeas, M. G. (2010). *Drag reduction by flow control on a car after body*. Int. J. Numer. Methods Fluids, 1222-1240.
10. W. Meile1 , G. Brenn1 , A. Reppenhagen2 , B. Lechner2 , A. Fuchs2C "Experiments and numerical simulations on the aerodynamics of the Ahmed body" CFD Letters, Vol. 3 (1) – March 2011.