VOLUME-8, ISSUE-1, JANUARY-2019 • PRINT ISSN No 2277 - 8160

Original Research Paper

Engineering

ANALYSIS OF AN AUTOMOBILE TO IMPROVE AERODYNAMICS

Anuj Dalvi B.E Mechanical SSJCOE Dombivli India,

Mihir Kathe* B.E Mechanical SSJCOE Dombivli India, *Corresponding Author

Shirodkar Prof. Of Mechanical Engineering SSJCOE Dombivli India,

ABSTRACT The Engineer is constantly conformed to the challenges of bringing ideas and design into reality. The Computational Fluid Dynamics is a step towards bringing numerous Hypothetical Concepts to reality, as the Analysis done gives better results compared to the Physical Analysis (Wind Tunnel Testing), without any costly setup and space requirement. In this Project, Modelling of Automobile (Car) was done using the Software-Solidworks. The Lift and the Drag of the car were determined by the Analysis of Fluid (Air) Flow around it using Software-Ansys Fluent. After that, with modifications like Air Vents, Rear Spoiler, the Analysis was repeated. Based on the values of Lift Force and Coefficient of Drag (Cd), optimal solution was considered.

KEYWORDS : cfd, drag, aerodynamic, spoiler, air vent,

INTRODUCTION

Aerodynamics is basically the science that deals with the study of movement of air and the way an object moves through the air.

The concept of Aerodynamics was brought into effect by the Wright Brothers in early 1900s by bringing man's dream of flying into reality. The concept of Aerodynamics has contributed hugely in all the types of Automobiles and the making of fighter planes, high speed trains, supercars and has been scrutinized to a different level.

Use of Aerodynamics through mathematical analysis, empirical approximations, wind tunnel experimentation and computer simulations, has formed a scientific basis for the ongoing developments.

The two major branches of Aerodynamics are –
(a) Incompressible Aerodynamics – It is concerned with the incompressible flow, which is the flow in which density is constant in both time and space.

The Subsonic (or the low-speed) flow dynamics is the further division of the incompressible Aerodynamics.

(b) Compressible Aerodynamics – It is concerned with the compressible flow, which is the flow in which the change of density with respect to pressure is non-zero along a streamline.

The further divisions of the compressible flow are –
(1) Transonic Flow – The term transonic refers to the range of flow velocities just below or above the local speed of sound i.e. 1225 km/h at standard sea level conditions corresponding to temperature of 15 degree Celsius.

(2) Supersonic Flow – The Aerodynamics is concerned with the flow speeds greater than the speed of sound.

DRAG COEFFICIENT

Drag Force is force acting opposite to the relative motion of an object moving with respect to the surrounding fluid. The Drag Force depends upon the velocity of the object. The Drag Force is directly proportional to the velocity, for laminar flow and to the squared velocity for turbulent flow. The Drag Force is denoted by (Fd) and calculated by using the equation-

$Fd = ½*\rho*v^2*Cd*A$
where, ρ - Density of Air
v- Relative velocity (object + surrounding fluid)
Cd – Coefficient of drag
A – Frontal Area of the vehicle

The coefficient of drag plays a vital role in the analysis of the Drag Force. It is proportional to the Drag Force and hence lesser value of coefficient reduces the resistive force.

OBJECTIVE:

This paper aims to accumulate all possible information & Knowledge of a Car's Aerodynamics focusing on the rear spoiler use.
1.Analysis of the air flow around the car without the rear spoiler

2.Analysis of the air flow around the car with a concept rear spoiler.

3.Effect of the Aerodynamics on the car and analysis with the variation of the rear spoilers on the down.

4.Estimating the Cd (Coefficient of Drift) on a high speed run of the car.

5.Comparison of the Cd values with (variations in rear spoiler designs) and without rear spoilers.

6.Drawing out the possible outcomes comparing the results & establishing the relation of using rear spoilers for better performance, reduced lift and drag.

THEORY:

CFD is the use of applied mathematics, physics and computational software to visualize how a gas or liquid flows as well as how the gas or liquid affects objects as it flows past. Computational fluid dynamics is based on the Navier-Stokes equations. These equations describe how the velocity, pressure, temperature, and density of a moving fluid are related.
CFD has been around since the early 20th century and many people are familiar with it as a tool for analyzing air flow around cars and aircraft. CFD has also become a useful tool in the data center for analyzing thermal properties and modeling air flow. CFD software requires information about the size, content and layout of the data center. It uses this information to create a 3D mathematical model on a grid that can be rotated and viewed from different angles.

PRE PROCESSING:

Vehicle Geometry Modeling:

For modeling the geometry, 3D modeling software Solidworks 2016 was used. The modeling process involved importing the vehicle blueprints into Solidworks with the help of which, 3D curves were projected. These curves then acted as boundaries to generate surfaces. The final surface model was converted into a solid part (refer Figure. 1) before importing it to Ansys.

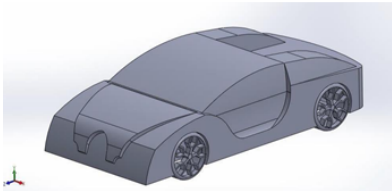

Figure1:

Creating fluid enclosure:

In order to simulate the air flow around the vehicle, a fluid volume needs to be created which will encompass the vehicle. This was done by creating an enclosure around the vehicle and subtracting the vehicle body. This enclosure acts as the air domain. To reduce the overall computational cost and time, the vehicle was considered symmetric laterally. The size of the enclosure was taken to be 30mm ahead of the car, 100mm above the car and beside the car 30mm spacing was left. Length of enclosure taken as 500mm and between the car rear and the end of the enclosure space 200mm was left.

Mesh genration:

While generating the mesh, sizing functions were used wherever necessary in order to obtain accurate lift/drag parameters. Two bodies of refinements were added to properly capture the flow in the region closest to the vehicle and also capture the flow in the wake. Since boundary layer separation has a significant effect on drag, First Aspect Ratio inflation were added to the vehicle surface to properly resolve the boundary layer. Minimum and maximum size of elements were taken as 1.8750mm and 500mm. The total number of elements obtained was 183218 and 35761 nodes.

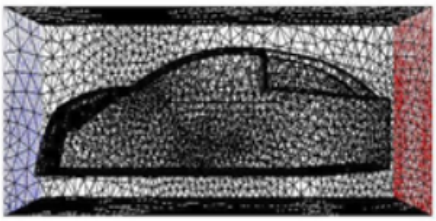

Figure 2: Mesh genrated without spoiler

SOLVER:

For this analysis, a pressure based steady state solver was used. The solution methods, equations used along with the input data are listed below:

- Least Squares Cell Based Solver.
- Realizable k - e p s i l o n m o d e l w i t h n o n - equilibrium wall functions.
- Air velocity at inlet: 350 m/s.
- Reference area to determine drag and lift coefficients – Frontal Area: 1 m2.
- Standard pressure consider for the analysis.
- Pressure Velocity Scheme was Simple.
- Momentum was Second Order Upwind.
- Under Relaxation Factors are as below; Pressure: 0.3
  Body forces: 1
  Density: 1
  Momentum: 0.7

SOLVER RESULT

The final solution was obtained by performing the iterations in three stages. First was initialization of Hybrid Solution with 35 iterations. Further Final solution was obtain in 100 iteration.

A final drag coefficient and lift of 0.456 and 0.5 were obtained.

PRE PROCESSOR- ANSYS CFX

The velocity, pressure contours, path lines and velocity vectors are shown below

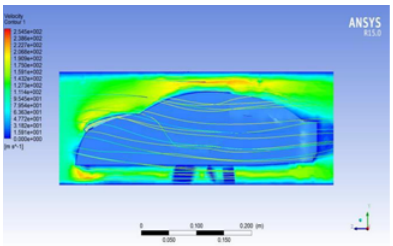

Figure 3: Velocity counter without spoiler.

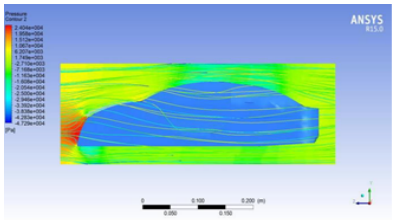

Figure 4: Pressure counter without spoiler

MODIFICATION OF GEOMETRY:

Addition of Spoiler, air vents, mirrors and alloy wheels

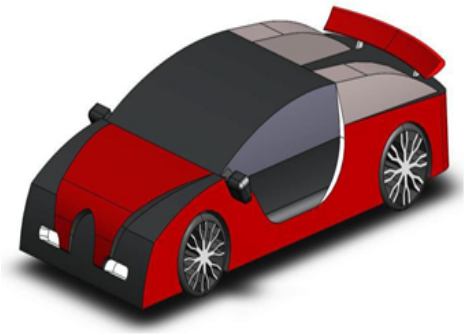

Figure 6: Modified 3D model

In order to improve the drag/downforce characteristics of the vehicle, the geometry was modified and a spoiler was added at the rear. The goal was to reduce the coefficient of drag and thus increase the efficiency of vehicle by providing streamline flow of fluid i.e. air over the surface of the car. Also to reduce the turbulent flow of the car at the read end by using spoiler.it also include study of effect of various aerodynamic efficiency affecting factors like air vents, mirrors and alloy wheels.

SOLVER RESULT- WITH SPOILER

The iterations were made in similar pattern as that of raw model. The drag and lift coefficients obtained from the spoiler are tabulated below.

Note that the difference in coefficients is with respect to the original model.

COMPARISON BETWEEN VALUES OF CD AND CL BEFORE AND AFTER ADDITION OF SPOILER.

MODEL	Cd	Cl	Cd	Cl
Raw model	0.456	0.5	0.094	0.014
Modified model	0.362	0.486		

Addition of a spoiler, air vents showed that there is decrease drag and a corresponding decrease in lift in all two cases.

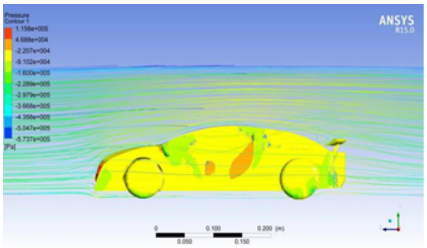

Figure 7: pressure stream after addition of spoiler.

TRANSIENT FLOW ANALYSIS

Transient flow is the flow, wherein, the flow velocity and Pressure are changing with time. When changes occur to a fluid systems such as during starting or stopping, in such a situation transient flow conditions exists. Otherwise the system is in steady state. Often, transient flow conditions persist as oscillating pressure and velocity waves for some time after the initial event that caused it.

In the laminar regime, we can assume that the velocity field does not vary with time, and get an accurate prediction of the flow behavior. As the flow begins to transition to turbulence, it is no longer possible to assume that the flow is invariant with time. Such problems can be solved by transient flow analysis which solves it by considering a time based domain. It will solve the problem in different time steps with variation of flow.

From the steady state result obtained, we see that the addition of spoiler considerably reduces the drag coefficient A transient flow analysis was therefore performed on that model to check the accuracy of the results.

SOLVER RESULT:

The final solution was obtained by performing the iterations in three stages. First was initialization of Hybrid Solution with 35 iterations. Further Final solution was obtain in 100 iteration.

A final drag coefficient and lift of 0.362 And 0.5 were obtained. Graph of lift and drag coefficient are as shown below;

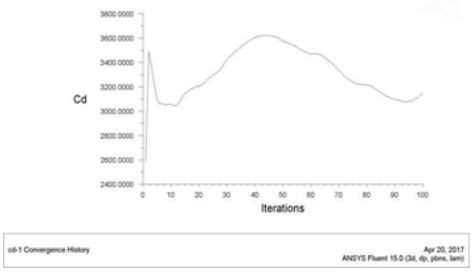

Figure 8: Coefficient of drag

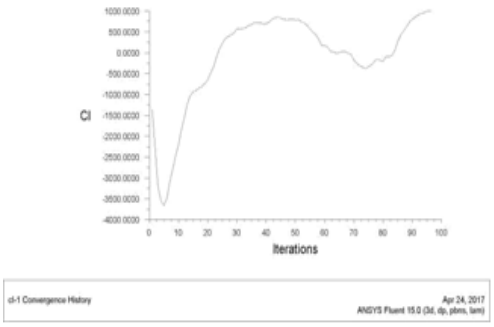

Figure 9: Coefficient of lift.

The effect of addition of spoiler on the flow of fluid i.e. air around the vehicle are shown below;

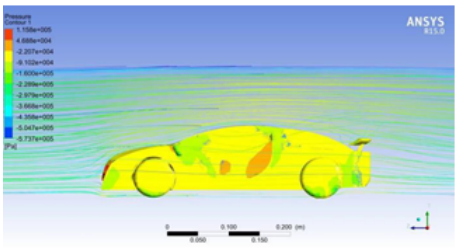

Figure 10: Velocity streamline after addition of spoiler.

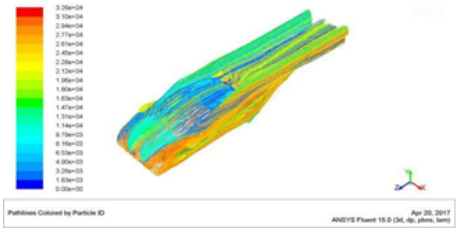

Figure 11: Path lines of vehicle with spoiler.

CONCLUSION:

Different types of cars use rear trunk spoilers. The research focused on the application of the rear spoiler designed in 2D and explained in CFD processing tools which address the problem statement by allowing the model car to withdraw its drag and reducing the lift. This has the effect of streamlining the model car to attain the lowest possible drag and lift when required in high velocity.

The addition of a spoiler helped reduce the lift considerably (0.094 reduction in Cl) while reducing the drag coefficient (0.014 decrease in Cd). Modifying the aerodynamic curve angle angle by 2.2 deg. led to a greater reduction in lift. However, the corresponding drag penalties were also higher. The results obtained by performing the steady state analysis were then confirmed by performing a transient simulation, both providing a similar value of Cd. The system was successfully simulated and compared against the performance considering two cases. The CFD simulation allowed a direct comparison of two cases. This comparison also helped in comparing the post process values obtained from ANSYS Fluent. The first set up was to analyze the background and possible scope of the product in the automobile industry.

REFERENCE:

[1] Team of Croatian Authors, "CFD Analysis of a concept car in order to improve Aerodynamics", University of Osijek, Croatia, 2011
[2] Amol Mangrulkar, "Aerodynamic Analysis of a Car Model using Ansys Fluent 14.5", Rajiv Gandhi Institute of Technology, 2014.
[3] Oleg Zikanov, "Essential Computational Fluid Dynamics", John Wiley & Sons, Inc. Hoboken, New Jersey, March 2010
[4] Mustafa Cakir, "CFD Study on aerodynamic effects of a rear wing/spoiler on passenger vehicle", Santa Clara University, 2012.
[5] Tomasz Janson, "Numerical Analysis of Aerodynamics characteristics of a High-speed car with movable Bodywork Elements", 2015
[6] Website: http://autospeed.com/cms/title_Aero-Testing-Part4/A_108676/article.html
[7] W. Seibert "CFD in Aerodynamic Design Process of Road and Race Cars",Fluent Deutschland GmbH, FLUENT Technical Notes TN155, Presented at European
[8] Automotive Congress, Bratislava, Slovakia, June 18-20 2001
[9] Klaus Gersten, E. Krause, H. Jr. Oertel, C. Mayes "Boundary-Layer Theory", Herrmann Schlichting, 8th Edition, Springer 2004
[10] W. Seibert, M. Lanfrit, B. Hupertz and L. Krüger "A Best- Practice for High Resolution Aerodynamic Simulation around a Production Car Shape" 4th MIRA International Vehicle Aerodynamics Conference, Warwick, UK, October 16-17, 2002
[11] Website: http://bugattipage.com/ride.htm
[12] Website: http://www.grc.nasa.gov/WWW/K-12/airplane/ boundlay.html