DESIGN OPTIMIZATION AND CFD ANALYSIS OF CAR USING ACTIVE MOUNTING TO REDUCE DREG COEFFICIENT AND REDUCE LIFT TO SLEEP OF CAR AT HIGHER SPEED

RAJYAGURU PRANAV 1,TANDEL HITESH 2,KISHAN VAGHELA 3,
123student-Vadodara Institute Of Engineering -Vadodara-India

Abstract: An analytical study on air flow which effects and resulting dynamics on the car is presented. The research deal with Computational Fluid Dynamic analysis and simulation to maximize down force and minimize drag during high speed of the car. Using ANSYS FLUNT software and mentoring provided by ANSYS , the results employs efficient discretization techniques and real loading conditions to study down force on rear wing of the car with drag generated by all active mounted surfaces. Wing and external surface under high velocity runs of the car are presented. Optimization of wing direct angle of attack and geometry modifications on surfaces of the car are performed to enhance down force and reduce drag for higher degree of stability and to control during operation. Moreover to ensure more stability active aerodynamic spoiler which can adjust its height and its angle of attack according to speed. So the aim of research to find best height at which maximum down force and air flow separate so that car body can get high level of stability . And another factor is angle of attack at which highest value of down force we get.

Keywords: ACTIVE AERODYNAMICS , SPOILER , DOWN FORCE , STABILITY , ANSYS ( FLUENT ) , CATIA V6

1. INTRODUCTION

1.1 Introduction to active aerodynamics

Active aerodynamics is a combination of a various -stage adjustable rear spoiler. The spoiler lip is protected. In stage two (Speed), after 120 km/h, rear spoiler are partially extended. This helps to ensure a considerable level of stability, a low drag coefficient and enables a high top speed rear spoilers are now fully extended. Also, in this position, the rear spoiler is tilted by up to 15 degrees.

1.2 COMPTUNIONAL FLUID DYNAMICS

Computational fluid dynamics is the transformation of a situation to the mathematical model with governing equation. This whole equation are looped with other equations which are depend on another equations and solved together however, this calculation is very tough by manual source so it is calculated by metleb, ansys, simscale and various cfd soft wear . In background in cfd the equations of mass, momentum and energy also for drag and lift called navier stock equation . so that cfd is called combination of computer science, mathematics, physics.in our research we don’t require any heat transfer or any thermal turm we only interested in air streamlines.

1.2 MODELLING

The 3-d model is done using blue print of Porsche 911 turbo .Automobile body have lots of sharp curves and edges. so that it is very difficult to make solid model . By using of surface modeling automotive body can easily done and results will be perfect and can be implemented in real word and also used by automobile company.
SOFTWARE USD IN MODELING

1. CREO 2.0
2. CATIA V5
3. NX NASTERAN

![Surface Model of Car](image1)

**FIG-1 SURFACE MODEL OF CAR**

2. MESHING

For this the 3D model with enclosure was imported, then the enclosure was given named selections into various parts like body, wall, inlet and outlet and the required meshing conditions were applied and meshing was done on the imported geometry. Meshing is an important step of whole analysis because results are mostly depends on number of elements and node as the number of elements are high than order of accuracy is high.

**2.1 Meshing Size**

<table>
<thead>
<tr>
<th>ELEMENTS</th>
<th>3667923</th>
</tr>
</thead>
<tbody>
<tr>
<td>Use NODES</td>
<td>659019</td>
</tr>
<tr>
<td>Minimum Size</td>
<td>17.23mm</td>
</tr>
<tr>
<td>Maximum Size</td>
<td>500mm</td>
</tr>
</tbody>
</table>

**TABLE-1 MESH SIZING**

![Mesh Generation](image2)

**FIG-2 MESH GENERATION**
3. VIRTUAL TUNNEL GENERATION

Virtual Tunnel is one enclosure in which car model is put in mid position. In real condition car is moving however, in ANSYS car is steady and air is pass through tunnel.

![Wind Tunnel Diagram](image)

**FIG - 3 WIND TUNNEL**

4. ANALYSIS

The analysis of the model with enclosure was done in ANSYS FLUENT 16.2. For this at first the mesh was checked and after the approval of mesh various analysis parameters like models, materials and boundary conditions were set.

4.1. Models

The model used for analysis was K-epsilon.

4.2 BOUNDARY CONDITION

<table>
<thead>
<tr>
<th>Inlet type</th>
<th>Velocity inlet</th>
</tr>
</thead>
<tbody>
<tr>
<td>Outlet type</td>
<td>Flow-split outlet</td>
</tr>
<tr>
<td>Top and side faces</td>
<td>Slip shear stress specification</td>
</tr>
<tr>
<td>Symmetry face</td>
<td>Symmetry</td>
</tr>
<tr>
<td>Ground face</td>
<td>Wall (with the same velocity as the flow)</td>
</tr>
<tr>
<td>Car faces</td>
<td>Wall</td>
</tr>
<tr>
<td>Pressure</td>
<td>Atmospheric</td>
</tr>
<tr>
<td>Turbulence Intensity</td>
<td>3 %</td>
</tr>
<tr>
<td>Turbulence Specification</td>
<td>Intensity + length scale</td>
</tr>
<tr>
<td>Turbulent Length Scale</td>
<td>10 mm</td>
</tr>
<tr>
<td>Turbulent Velocity Scale</td>
<td>1 m/s</td>
</tr>
<tr>
<td>Velocity</td>
<td>40 m/s</td>
</tr>
</tbody>
</table>

**TABLE - 3 BOUNDARY CONDITION**

5. RESULTS AND DISCUSSIONS

Results are last stage but an important one. geometry of our research is too big in volume so the computation from all numerical value is become too complex and time consuming. so we try to do this based on color contour however it is the second form of numerical values.

In our research we had target to complete analysis at five angle of attack and at three speed so whole procedure we can not present here and we presented limited and important results.

Using ANSYS we complete analysis in two parts

(1) analysis of car body without spoiler at various speed

(2) analysis of car body using spoiler at various angle of attack and at various height

5.1 PRESSURE COUNTER AT 50M/S

In automotive analysis due to large number of elements and volume it is very complex to take every mathematical data. To save time and reduce complexity comparison of color counter is a way to compare down force.

FIG-4 EFFECT OF BACK PRESSURE AND IRREGULAR STREAM LINES

FIG-5 EFFECT OF DIFFUSER

FIG-6 STREAM LINES OVER CAR BODY
FIG-7 PRESSURE CONTOUR AT 50M/S WITHOUT SPOILER

FIG-8 PRESSURE CONTOUR AT 70M/S

FIG-9 GRAPH OF RESULT OF DREG AND LIFT AT 35M/S, 50M/S, 70M/S
6. CONCLUSION

In our research we had lots of analysis about 20-30 so it become very complex to conclude best angle attack for any speed moreover speed such as 70-80 m/s is very higher speed so that analysis should be very exact. At some stage such as 30-40 m/s we required only slip don't require too much down force. In addition to that we can't focus only on down force because at some value down force is high but drag is also high that is why we cannot use that angle at particular angle.

At the very high speed down force is very crucial turn also it required to check the streamlines, that should not creat the back pressure and negative pressure. cfd is definitely a best way to see aerodynamics however it is too complex and too much calculation is required with higher order of accuracy so that it can be applied.

Although the main goal was to pick the highest value of Lift forces to Drag forces ratio, the absolute values of the lift forces and drag forces also had to be considered. For example, the actual lift was the least of all angle-of-attack orientations. Hence the end goal was to identify the highest lift possible and then utilize the ratios as a method of optimization (see Table 3).

Here in table some of the best results are shown here we can not show all the combination here some of best result as per our goal is shown here. on this results active spoiler is designed by use of micro controller.
<table>
<thead>
<tr>
<th>Lift:DragRatio</th>
<th>5(degree)</th>
<th>6(degree)</th>
<th>8(degree)</th>
<th>13 degree</th>
</tr>
</thead>
<tbody>
<tr>
<td>150km/hr</td>
<td>8.92</td>
<td>8.87</td>
<td>9.02</td>
<td>9.12</td>
</tr>
<tr>
<td>200km/hr</td>
<td>9.06</td>
<td>8.31</td>
<td>7.76</td>
<td>8.91</td>
</tr>
<tr>
<td>250km/hr</td>
<td>7.8</td>
<td>7.2</td>
<td>8.32</td>
<td>8.94</td>
</tr>
</tbody>
</table>

TABLE -4 DATA FOR ACTICE SPOILER

7. REFERENCE


18. Website: [http://bugattipage.com/ride.htm](http://bugattipage.com/ride.htm)
