A CFD Investigation into the Transient Aerodynamic Forces Induced by Passing Maneuvers of a Truck and a Car

Mr. Harsh V. Patel, CGPIT  
Mr. Darshan A. Kapadia, CGPIT

Abstract - The transient aerodynamic forces occurring when one road vehicle overtakes another were investigated using two-dimensional (2D) computational fluid dynamics. A non-dimensional number Drag coefficient C_d is most important for entire discipline. Performance, fuel economy, emission and top speed are important attributes of the vehicle and they all are influenced by drag coefficient. Vehicle models used for this study are a truck and a car. A numerical simulation of flow around vehicles at different speeds and at different positions of the vehicles is performed using commercial fluid dynamic software ANSYS FLUENT. The study focuses on CFD based drag prediction on the car and truck body. A two-dimensional computer model of a car and truck is used as the base model in this study which is generated in the commercial software ANSYS WORKBENCH. In this study, six different positions of car in y-direction are carried out. Also, in this study air flows over the vehicle models at three different speeds 40 kmph, 60 kmph, and 80 kmph. The flow model used for this study is k-epsilon realizable. From this analysis, it is observed that when car overtakes truck yawing force is acting on the car when it goes away from truck and at that time sometimes it changes to turn a car.

Keywords - Passing maneuvers, Aerodynamic forces, CFD Analysis, Overtaking vehicles.

I. INTRODUCTION

Aerodynamics is a branch of fluids mechanics which study about the forces generated on a body in a flow. The aerodynamics usually involves calculation in the properties of the flow such as pressure, velocity, temperature, density as a function of space and time. In order to calculate or approximate the forces and moments acting on the bodies in the flow, we must understand the pattern of the flows.[1]

As one vehicle passes another during an overtaking manoeuvre the flow fields around the two vehicles interact generating transient aerodynamic forces. These forces can have an adverse effect on vehicle handling and stability. Practical problems regarding the relative movement of wind tunnel models and data acquisition under unsteady conditions have meant wind tunnel simulations of overtaking maneuvers have traditionally been completed using the quasi-steady approach.

When the vehicle is moving at a considerable speed, there are several forces are applied to vehicle in different directions. Figure 1.3 shows the details sketch view of the various forces acting on the vehicle body. As shown in the free body diagram below, there are six forces acting on the vehicle:

1. Rolling Resistance,
2. Drag,
3. Lift,
4. Gravity,
5. Normal,

![Figure 1 Forces acting on car body][2]

Rolling resistance force is due the tires deforming when contacting the surface of a road and varies depending on the surface being driven on. The normal force is the force exerted by the road on the vehicle's tires. Lift force acting on the vehicle body vertically. This force causes the vehicle to get lifted in air as applied in the positive direction, whereas it can result in excessive wheel down force if it is applied in negative direction. Drag force is the force acting on the vehicle body resisting its forward motion. This force is an important force to be considered while designing the external body of the vehicle, since it covers about 65% of the total force acting on the complete body.

II. LITERATURE REVIEW

R. J. Corin et. Al.[3] The transient aerodynamic forces occurring when one road vehicle overtakes another were investigated using two-dimensional (2D) computational fluid dynamics. The relative velocity of the vehicles was varied to allow comparison of the quasi-steady and unsteady modeling approaches. The quasi-steady approach captured the pseudo-periodic variation in the aerodynamic forces during the overtaking maneuver. However, at vehicle velocities typical of motorway driving conditions, it did not adequately predict the magnitude of these forces. The study also identified significant dynamic flow features occurring during overtaking maneuvers in a crosswind. The dynamic variation in the aerodynamic forces was up to 400% greater than that predicted using quasi-steady analysis, indicating that the quasi-steady approach is totally unsuitable for modeling overtaking maneuvers in a crosswind. These dynamic effects are likely to have a considerable impact on the stability of the vehicles involved. With little existing work on passing maneuvers in a crosswind, these results highlight the importance of dynamic effects and the need for further investigation into the problem.

Tank Nilesh R et. al.[4] External aerodynamic analysis is considerable when any solid body obstructs the flow, like on road vehicle. Generally flow is parallel to the vehicle direction and in opposite, but in some cases like on high way where, wind is blowing from sides also. It is more interesting if the vehicles are overtaking each other, because considerable transient condition is occurring. For overtaking vehicles transient condition is inevitable which has considerable effects on results. Numerical flow around the passenger car, a bluff body, is the subject of present work in both, steady state as well as transient condition. Cross wind situation will arise transient
condition on highways frequently which can affect the directional stability of the vehicle. Here 2D cross flow and axial flow steady as well as transient cross flow with dynamic mesh CFD simulation is completed using commercial package. Generally numerical analysis predicts values near to experimental value, which is in terms of time and cost saving.

III. METHOD AND MODELS

A. NUMERICAL METHOD

CFD codes are structured around the numerical algorithms that can handle fluid flow problems. All the CFD commercial packages available in the market have three basic elements, which divide the complete analysis of the numerical experiment to be performed on the specific domain or geometry. The three basic elements are:-

1. Pre-Processor,
2. Solver,
3. Post-processor.

B. GENERIC MODELS

To construct geometry in ANSYS Workbench, dimensions of generic model of a car and a truck are required. For getting the dimension from generic model, “GET DATA” software is used.

![Figure 2 Dimension of Car and Truck](image)

Dimensions of a car are as follow:
- Length of a car = 4910 mm,
- Width of a car = 1891 mm.

Dimensions of a truck are as follow:
- Length of a truck = 16084 mm,
- Width of a truck = 2500 mm.

![Figure 3 Dimension of outer surface](image)

Length of this domain is set as 112588 mm and the width of this domain is set as 69977 mm.
IV. NUMERICAL SIMULATION

A. CAD MODEL

The model of a passenger car and a truck has been generated with the help of commercial package ANSYS WORKBENCH. In figure 4-1 and 4-2, it can be seen that there is no proper dimensions of all the points of the model. The geometry is much complicated than some of the assumptions are taken. Car geometry is much complicated than truck body. Car body is curved shape from both the sides, whereas Truck body is curved from front and simple from rear. With help of commercial package software “GET DATA”, dimensions of all points are derived to draw geometry. 2D model is generated in the commercial package ANSYS WORKBENCH which is shown in below Figure 5-1.

![Figure 4 Virtual air-box of Geometry](image)

B. MESH GENERATION

The triangular shape surface mesh was used due to its proximity to changing curves and bends. These elements easily adjust to the complex bodies used in changing curves and aerospace bodies. With the help of commercial package ANSYS WORKBENCH the mesh is generated, in the size function input the required size limit than after with use of face mesh tool chooses the triangular mesh than generate mesh in the virtual air-box of the geometry which illustrate in Figure 5-2 and 5-3.

![Figure 5 Mesh generation](image)

Total No. of Elements = 19450  
Total No. of Nodes = 39515  
Minimum Skewness = 8.1478*10^{-07}  
Maximum Skewness = 0.55804  
Average Skewness = 4.4222*10^{-02}
C. BOUNDARY CONDITIONS

<table>
<thead>
<tr>
<th>Boundary Name</th>
<th>Boundary Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet</td>
<td>Velocity inlet</td>
</tr>
<tr>
<td>Outlet</td>
<td>Pressure outlet</td>
</tr>
<tr>
<td>Symmetry Up</td>
<td>Symmetry</td>
</tr>
<tr>
<td>Symmetry Down</td>
<td>Symmetry</td>
</tr>
<tr>
<td>Car</td>
<td>Wall</td>
</tr>
<tr>
<td>Truck</td>
<td>Wall</td>
</tr>
<tr>
<td>Interior Area</td>
<td>Fluid</td>
</tr>
</tbody>
</table>

D. SIMULATION IN ANSYS FLUENT

For the simulation of geometry, commercial package software ANSYS FLUENT is used. In this analysis, the k-epsilon realizable model is chosen. There are different types of analysis done at the different position of car from truck and at the different speeds of 40 km/hr, 60 km/hr, and 80 km/hr. Here only considered the frontal drag which is more affected than other skin drag, friction drag, and pressure drag. Our aim is to find out the coefficient of drag for a car and a truck.

Following are six cases which we solve in commercial package of ANSYS FLUENT.

1. CASE - 1 : $\Delta Y/L = 0$,
2. CASE - 2 : $\Delta Y/L = -0.5$,
3. CASE - 3 : $\Delta Y/L = -1.0$,
4. CASE - 4 : $\Delta Y/L = +0.5$,
5. CASE - 5 : $\Delta Y/L = +1.0$,
6. CASE - 6 : $\Delta Y/L = +1.5$.

Where, $\Delta Y =$ Distance between front of a truck and a car in X-direction $L =$ Length of a truck.

V. RESULTS OF SIMULATION

A. CASE - 1 : $\Delta Y/L = 0$

![Figure 6 Geometry and Meshing of Case-1](image)
Geometry and meshing of case 1 are shown in Figure 6-1. Maximum pressure and maximum velocity are observed 79.1 Pascal and 20.2 m/s respectively. Velocity contour is also shown in Figure 6-3. Reverse flow is observed behind the truck.

B. RESULTS FOR SPEED 40 KMPH.

<table>
<thead>
<tr>
<th>Sr. No.</th>
<th>Case</th>
<th>Co-efficient of Drag for Car ($C_d1$)</th>
<th>Co-efficient of Drag for Truck ($C_d2$)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.</td>
<td>$\Delta Y/L = 0$</td>
<td>0.70127</td>
<td>1.18530</td>
</tr>
<tr>
<td>2.</td>
<td>$\Delta Y/L = -0.5$</td>
<td>0.38713</td>
<td>1.24021</td>
</tr>
<tr>
<td>3.</td>
<td>$\Delta Y/L = -1.0$</td>
<td>0.38620</td>
<td>1.29940</td>
</tr>
<tr>
<td>4.</td>
<td>$\Delta Y/L = +0.5$</td>
<td>0.58418</td>
<td>1.34030</td>
</tr>
<tr>
<td>5.</td>
<td>$\Delta Y/L = +1.0$</td>
<td>0.24083</td>
<td>2.02332</td>
</tr>
<tr>
<td>6.</td>
<td>$\Delta Y/L = +1.5$</td>
<td>0.37865</td>
<td>1.31310</td>
</tr>
</tbody>
</table>
C. RESULTS FOR SPEED 60 KMPH.

<table>
<thead>
<tr>
<th>Sr. No.</th>
<th>Case</th>
<th>Co-efficient of Drag for Car (Cd1)</th>
<th>Co-efficient of Drag for Truck (Cd2)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.</td>
<td>∆Y/L = 0</td>
<td>0.69214</td>
<td>1.1471</td>
</tr>
<tr>
<td>2.</td>
<td>∆Y/L = -0.5</td>
<td>0.38196</td>
<td>1.2295</td>
</tr>
<tr>
<td>3.</td>
<td>∆Y/L = -1.0</td>
<td>0.37780</td>
<td>1.2893</td>
</tr>
<tr>
<td>4.</td>
<td>∆Y/L = +0.5</td>
<td>0.57587</td>
<td>1.3134</td>
</tr>
<tr>
<td>5.</td>
<td>∆Y/L = +1.0</td>
<td>0.23708</td>
<td>2.0075</td>
</tr>
<tr>
<td>6.</td>
<td>∆Y/L = +1.5</td>
<td>0.37188</td>
<td>1.3011</td>
</tr>
</tbody>
</table>

D. RESULTS FOR SPEED 80 KMPH

<table>
<thead>
<tr>
<th>Sr. No.</th>
<th>Case</th>
<th>Co-efficient of Drag for Car (Cd1)</th>
<th>Co-efficient of Drag for Truck (Cd2)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.</td>
<td>∆Y/L = 0</td>
<td>0.68697</td>
<td>1.1240</td>
</tr>
<tr>
<td>2.</td>
<td>∆Y/L = -0.5</td>
<td>0.37711</td>
<td>1.2160</td>
</tr>
<tr>
<td>3.</td>
<td>∆Y/L = -1.0</td>
<td>0.37258</td>
<td>1.2809</td>
</tr>
<tr>
<td>4.</td>
<td>∆Y/L = +0.5</td>
<td>0.57190</td>
<td>1.2883</td>
</tr>
<tr>
<td>5.</td>
<td>∆Y/L = +1.0</td>
<td>0.23260</td>
<td>1.9982</td>
</tr>
<tr>
<td>6.</td>
<td>∆Y/L = +1.5</td>
<td>0.36704</td>
<td>1.2915</td>
</tr>
</tbody>
</table>

E. COMPARISION

Calculation the values of Cd1 and Cd2 for six different cases are done at three different speeds. From all that data, graph of Cd vs. ∆Y/L can be drawn to compare the values of drag at different speeds and for different cases. The graph is shown in graph 6-1.

Graph 1 Comparisons
VI. Conclusion

From above graph and results of all cases, the conclusions are as follows:

- At ∆Y/L = 0 the value of co-efficient of drag for truck (Cd2) is lowest from all the cases and the value of co-efficient of drag for Car (Cd1) is highest from all the cases. Reason behind it is that the car and truck are at same position means besides each other so that pressure effect is highest on car.

- At ∆Y/L = +1 the value of co-efficient of drag for truck (Cd2) is highest from all the cases and the value of co-efficient of drag for car (Cd1) is lowest from all the cases. Reason behind it is that the pressure on truck is higher so the effect of car is on truck but there is no lower effect of truck on car.

- At ∆Y/L = +1 i.e. when car comes towards the truck and starts overtake it the pressure is highest at the below of the truck and at ∆Y/L = -1 i.e. car leaves the truck the pressure is highest at the upper of the truck.

VII. References


