Research Journal of Recent Sciences ISSN 2277-2502
Vol. **1(12),** 55-63, December **(2012)** *Res.J.Recent Sci.*

Numerical Simulation of Aerodynamic forces acting on Passenger Vehicle While Overtaking

Tank Nilesh R[1] and R. Thundil Karuppa Raj[2]
School of Mechanical and Building Sciences, VIT University, Vellore-632014, TN, INDIA

Available online at: www.isca.in
Received 16th August 2012, revised 31st October 2012, accepted 5th November 2012

Abstract

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 high ways 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.

Keywords: overtaking, aerodynamic, bluff body, wake region, dynamic mesh, CFD.

Introduction

Vehicle stability and fuel economy are major parameters for good auto-mobile design. During axial flow vehicle stability will not be issue, because the stagnation point will be near to the middle of front part. Vehicle is more stable if the geometric centre, centre of gravity and stagnation point should be fall in line, and also GC (geometric centre), CG (centre of gravity), and moment centre should be as nearer as possible[1]. While vehicles are overtaking due to un-steady condition pressure distribution will be different compare to steady state. This situation can be worst if vehicle is emerged in cross wind, because in cross wind stagnation point will shift towards from where the flow is coming. That can be revealing from contours. In present work drag, side force and moment force co-efficient for 2D case are measured, and compare between two vehicles. Significant changes can be observed from the plots, from $\Delta y/L= -0.5$ to $\Delta y/L= +0.5$. It reveals that there are a considerable changes happening at the interaction of two vehicles.

The idea of vehicle aerodynamic is borrowed from ship and air craft design. The difference between aircraft design and vehicle aerodynamic is unsuitability of lift force and boundary layer separation present in vehicles. In vehicle design, flow separation can be seen at the rear end whereas it will be attached in aircraft design.

CFD analysis of Okumura and Kuriyama concluded that dynamic effects are important and yawing moment is 100% higher than steady state results at ratio $Vr / V= 0.5$[2].

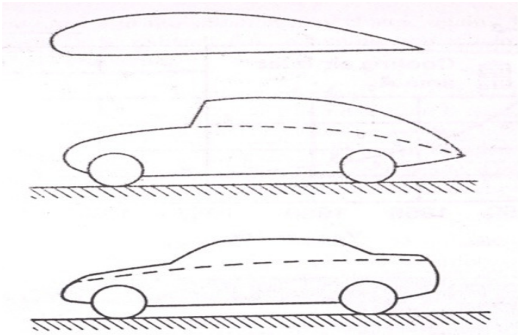
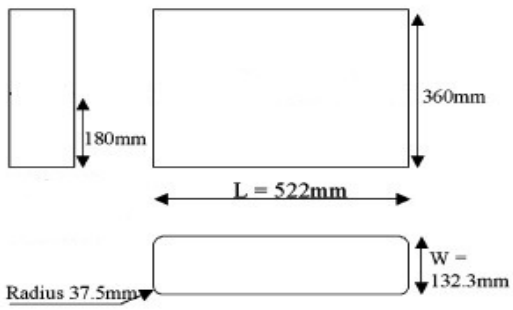


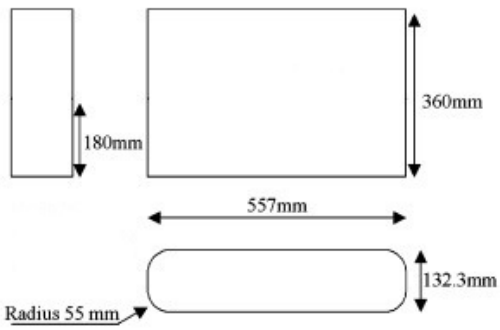
Figure 1[1]
Shape optimization

Experimental and CFD study of Gillieron and Noger with different range of relative velocity, different free stream velocity and scaled Ahmed model came to conclusion that side force is 60-120% higher in un-steady case than predicted in steady state condition. Results of Noger et al depicts that aerodynamic forces on overtaking and overtaken vehicles are velocity dependent when the ratio Vr / V is greater than 0.2[3].

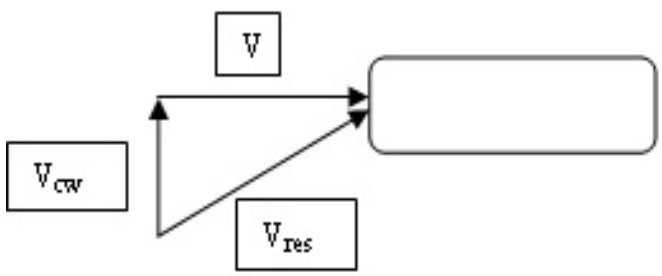
It is broadly accepted that cross wind effect can adversely influence vehicle handling and stability. Stagnation pressure is around 21% exceeds then steady state condition and after overtaking the flow separation re-establishment is also not present for transient case, like in steady state. Vehicle attributes will also affect the constant values as well as flow separation[4]. Also these results may vary due to geometry changes, different velocity ranges and angle at which flow is coming. Here little wok is presented to understand the flow phenomenon around overtaking vehicles for particular parameters.



(a)



(b)



(c)

Figure-2
a) and b) Dimensions of model 1 and 2 respectively; (c) lateral and longitudinal spacing (All dimensions are in mm)

Non-Dimensional Co-Efficient: Non-dimensional co-efficient are find out from standard equations (1)-(4) expressed below

$$C_p(s) = \frac{P_{(s)} - P_\infty}{1/2\rho V_{res}^{\ 2}} \quad (1)$$

$$C_d = \frac{D/span}{1/2\rho V_{res}^{\ 2} W} \quad (2)$$

$$C_s = \frac{SF/span}{1/2\rho V_{res}^{\ 2} L} \quad (3)$$

$$C_{ym} = \frac{M/span}{1/2\rho V_{res}^{\ 2} L^2} \quad (4)^5$$

Methodology

Computational Mesh: Computational domain size was set as 2L each side of vehicle and 3L for upper and downside. This domain is hired, that is used by Okumura and Kuriyama. Mesh was generated with commercial package ANSYS ICEM CFD. To predict the flow near wall spacing was 0.3 and enlargement ratio kept as 1.1. Number of mesh element is enough to solve the case with this set up. Also one case was tried with fine mesh with different set up; there is not much significant difference. Increasing the mesh size will improve the accuracy and also cost of simulation. With optimum size also can get the results with not much significant results, which will also decrease the cost as well.

Dynamic Mesh: Mesh is controlled by three methods: i. Smoothing, ii. Layering, iii. Remeshing. Considering the first cell height and largest, control parameters can adjust which will governs the merging of nodes and remeshing. Two types of smoothing are available spring smoothing and laplacian smoothing mainly use for tetra/tria while layering frequently use for hexa/quad if boundary motion is only present. Special features are applicable according to 2d or 3d[6].

Boundary Condition: Boundary conditions are specified properly in FLUENT to capture the real condition of flow. In general for axial flow external aerodynamic cases only inlet and outlet conditions are predominant while side walls are defining as 'wall' only but in cross flow it is mandatory to keep side as inlet and outlet also, velocity is mentioned in terms of components, such a way that resultant will be at 20º to the vehicle with respect to lateral direction. Freestream velocity is, V=27 m/s, cross-wind velocity is V_{cw} = 9.8 m/s. Based on these, resultant V_{res} can find out which is used for calculation of co-efficient. For transient condition inevitable feature are algorithm, based on selection of it, control parameter, time stepping method, time step and numbers of it.

Turbulence Modelling: One equation Spalarat-Allmaras model is used. It is effectively a low-Reynolds-number model, requiring the viscous influenced region of the boundary layer to be properly resolved. Near wall the gradient of transported variable i.e. kinematic viscosity in the model is much less compare to k- ε or k-ω models, so it is less sensitive to numerical error when non-layered meshes are used near walls[6]. Further only one equation is to be solved so computation time and cost is less. But model is often criticized for their inability to rapidly accommodate changes in length scale. Without clearly defined geometric-separation point, problem can be more complicated. The point of separation is strongly dependent on the flow structures and turbulence production within the buffer region of the boundary layer therefore detail turbulence modeling is required to predict separation as well as vertices formation in wake region which is predominant for drag[7]. These vertices are the energy consumer, taking the energy from main flow and forming large eddies gathering small eddies, which is in result weaken the flow.

Solution Procedure: Equations which are governing the flow are

Continuity Equation[8]

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0$$

Momentum Equations

$$\rho\frac{Du}{Dt}=-\frac{\partial p}{\partial x}+\frac{\partial \tau_{xx}}{\partial x}+\frac{\partial \tau_{yx}}{\partial y}+\frac{\partial \tau_{zx}}{\partial z}+\rho f_x$$

$$\rho\frac{Dv}{Dt}=-\frac{\partial p}{\partial y}+\frac{\partial \tau_{xy}}{\partial x}+\frac{\partial \tau_{yy}}{\partial y}+\frac{\partial \tau_{zy}}{\partial z}+\rho f_y$$

$$\rho\frac{Dw}{Dt}=-\frac{\partial p}{\partial z}+\frac{\partial \tau_{zx}}{\partial x}+\frac{\partial \tau_{zy}}{\partial y}+\frac{\partial \tau_{zz}}{\partial z}+\rho f_z$$ [8]

Standard solution procedure is followed. From moment equation pressure term solves by different methods depends on the type of solution, e.g. for steady state SIMPLE pressure-velocity coupling scheme can use while for simple transient SIMPLEC can use. PISO can maintain a stable calculation with a larger time step and an under-relaxation factor of 1.0 for both momentum and pressure. It is preferable to use Coupled scheme for poor mesh and large time step. Different discretization schemes are also available, for few iterations it is suitable to run the case in first order upwind scheme, after that second order is preferable, to control numerical diffusion, because in first order, diffusion will be more which lead to unstable solution. Upwind schemes are designed to numerically simulate more properly the direction of propagation of information in a flow field along the characteristic curve. As a result if upwinding scheme is used in proper manner calculation of sharp discontinuities can possible with no oscillation. Flow type is considered as inviscid, incompressible flow which is governed by elliptical partial differential equation, and also relaxation techniques, essential for iterative procedure are classic numerical methods to solve elliptical problems[8]. The structural analysis of the over taking vehicles can be carried out numerically similar to the literatures[9,10]. A number of researches using commercial CFD tool for automotive applications are cited in the literatures[11-15].

Results and Discussion

Cross-Wind Flow: Steady state, pressure variation along the model in a 20° crosses wind. Lateral Spacing Lateral spacing $\Delta x/W$ = 0.5, V=27 m/s, V_{cw} = 9.8 m/s, R_e No = 1e6. V_{res} = 28.7235 m/s. Left-hand model is overtaken and right-hand model is overtaking (figure-3).

From the contours it reveals that before overtaking, i.e (figure a) vehicle_2 has pressure developed at left-side, as it keep on moving stagnation pressure is reducing upto certain extent (figure b and c) due to the inluence of wake region of vehicle_1, and from fig 'd' to 'g' pressure of vechicle_2 is continuously increasing, and after figure 'g' it will start to stabilize towards the normal pressure.

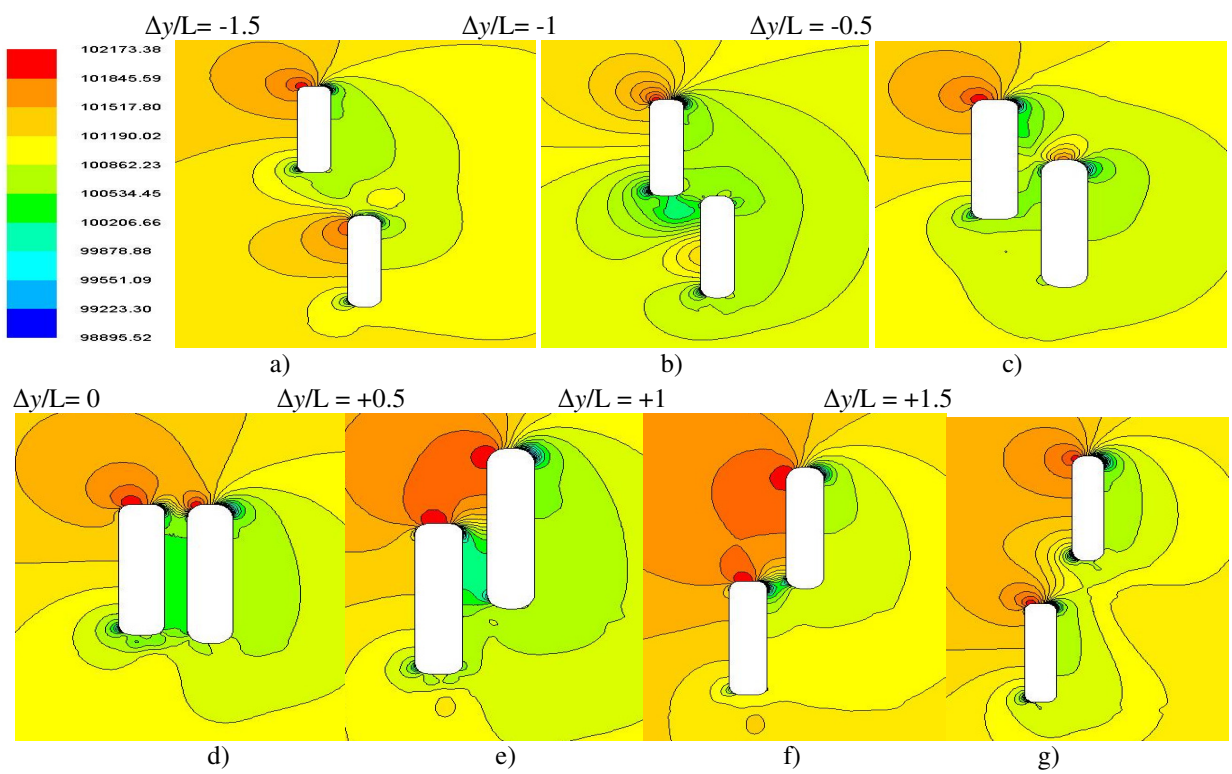


Figure 3
Absolute pressure contours of 20° degree cross flow condition

Flow Separation, Suppression and Re-Establishment: It seems from the plot, flow separation suppression and re-establishment is happening at particular position, i.e. while overtaking the effect is present of relative velocity both model, when vehicle_2 is about to reach to vehicle_1, i.e. at Δy= -0.5 the suppression started and where both model were parallel reveals that flow is totally suppressed, and again after Δy= 0 separation re-establishment started.

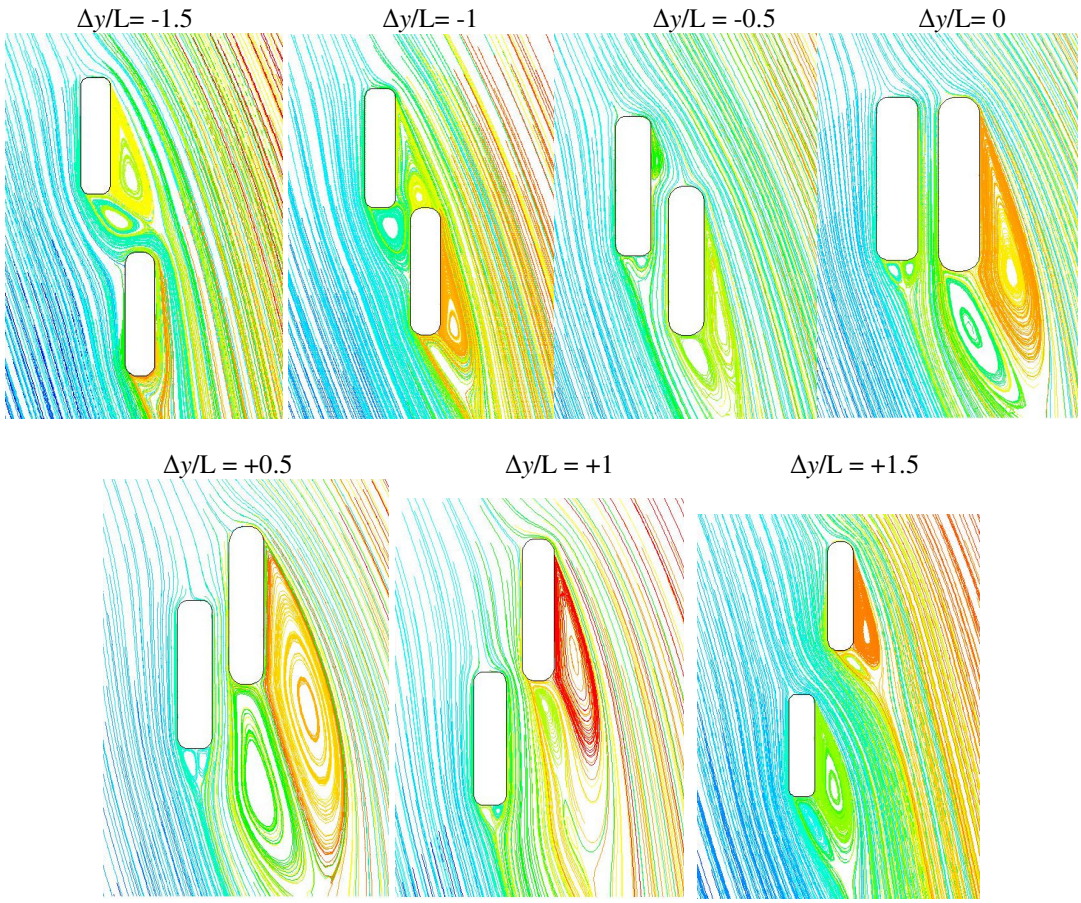


Figure 4
Pathlines for 20° degree cross flow condition

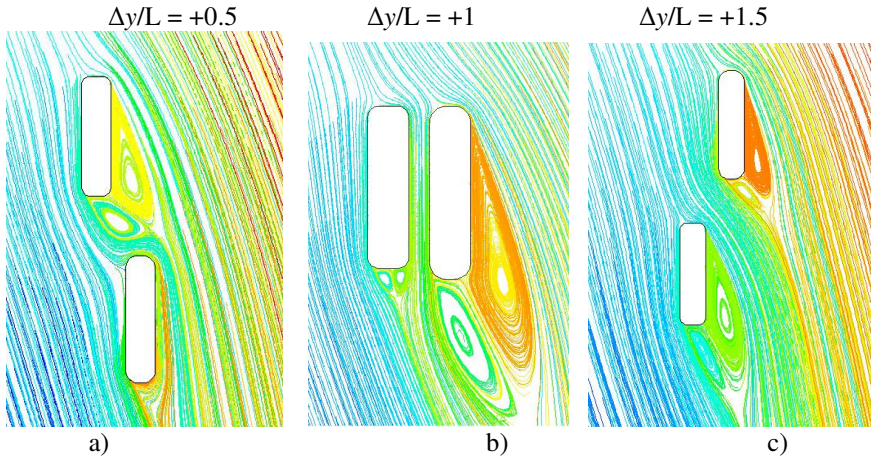


Figure 5
a) Flow separation, b) suppression and c) re-establishment

Axial Flow Results: Pressure Contours: Steady state solution, for axial (0˚ cross wind) flow, Lateral Spacing $\Delta x/W$ = 0.5, V=27 m/s, R_e No = 1e6.

Pressure contours presented over here clearly indicates that in axial flow there will not be significant deviation in stagnation pressure. Regarding the position of the vehicles there will not much effects of vehicle_1 on vehicle_2 or vice versa. Finally in axial flow either only one vehicle motion or overtaking motion, it will not affects more compare to cross flow, and also this effect depends on the lateral spacing.

Figures depicted that directional stability is not issue for axial flow around the vehicles because there is no critical parameter like yawing moment that creates the direction instability also no flow separation, suppression and re-establishment present. From the pressure and streamline contours it reveals that there is no effects of wake region of vehicle_1 on vehicle_2.

Absolute Pressure Contours: Variation in pressure distribution exists along the length scale which is clearly indicating the difference between steady state and transient condition. Fig a. to g. flow is strengthened at the wake region of the moving vehicle which is not captured in steady state. From dynamic pressure contours it reveals.

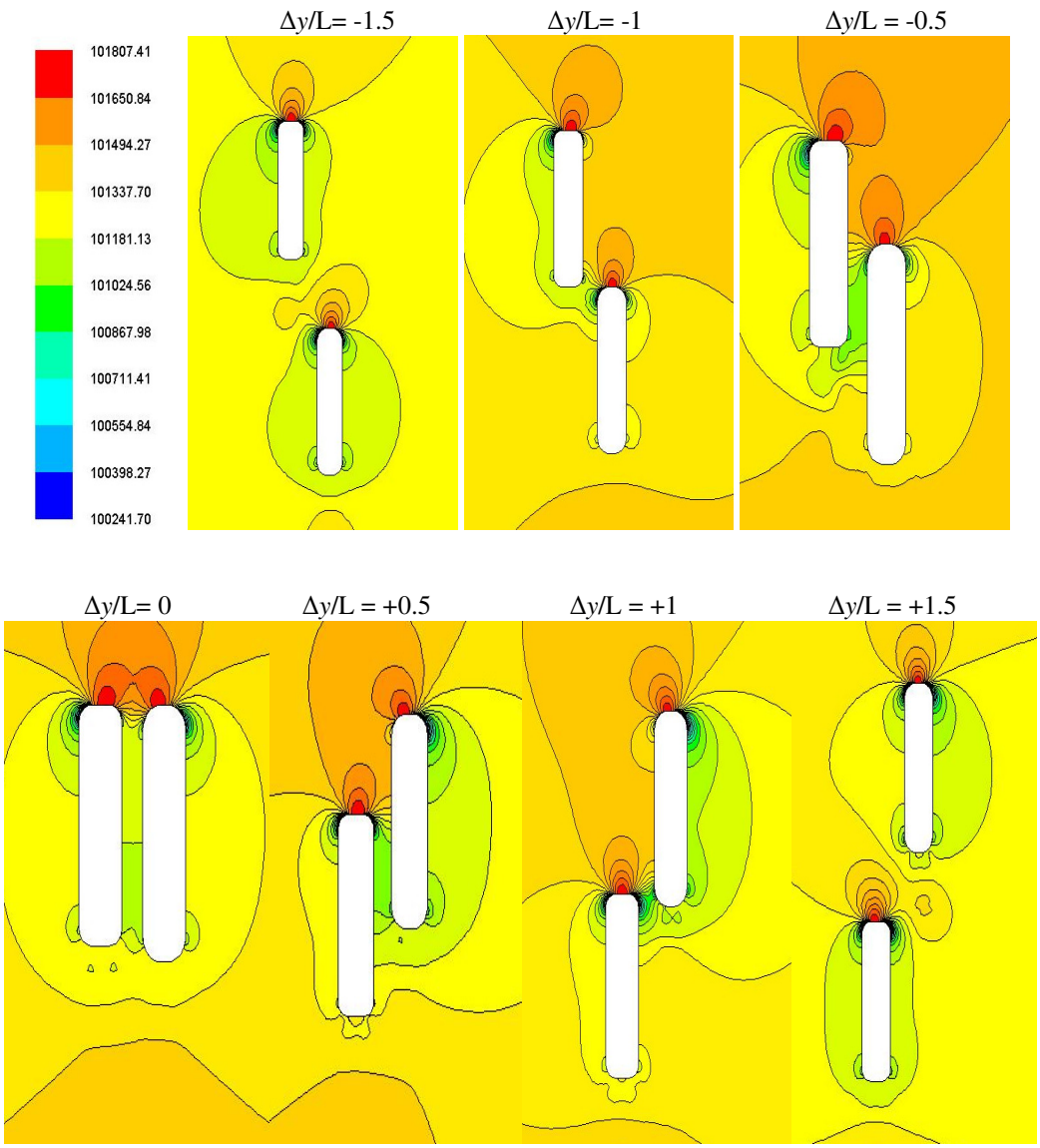


Figure 6
Absolute pressure contours of axial flow

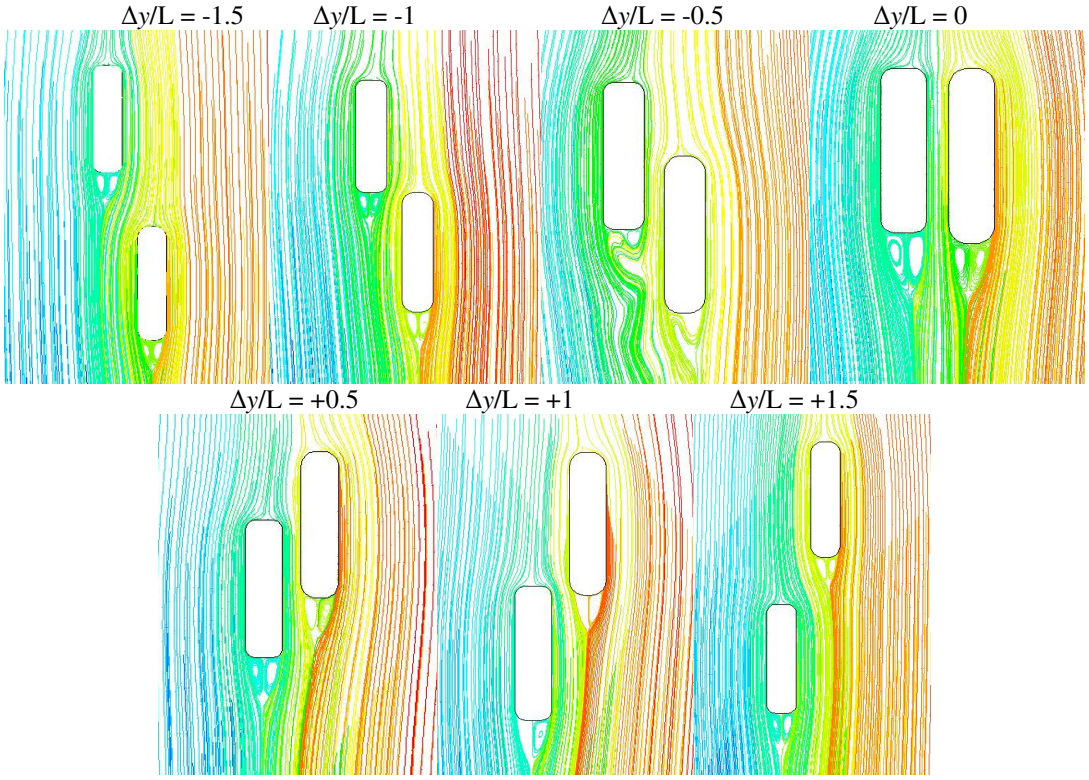


Figure 7
Pathines of axial flow

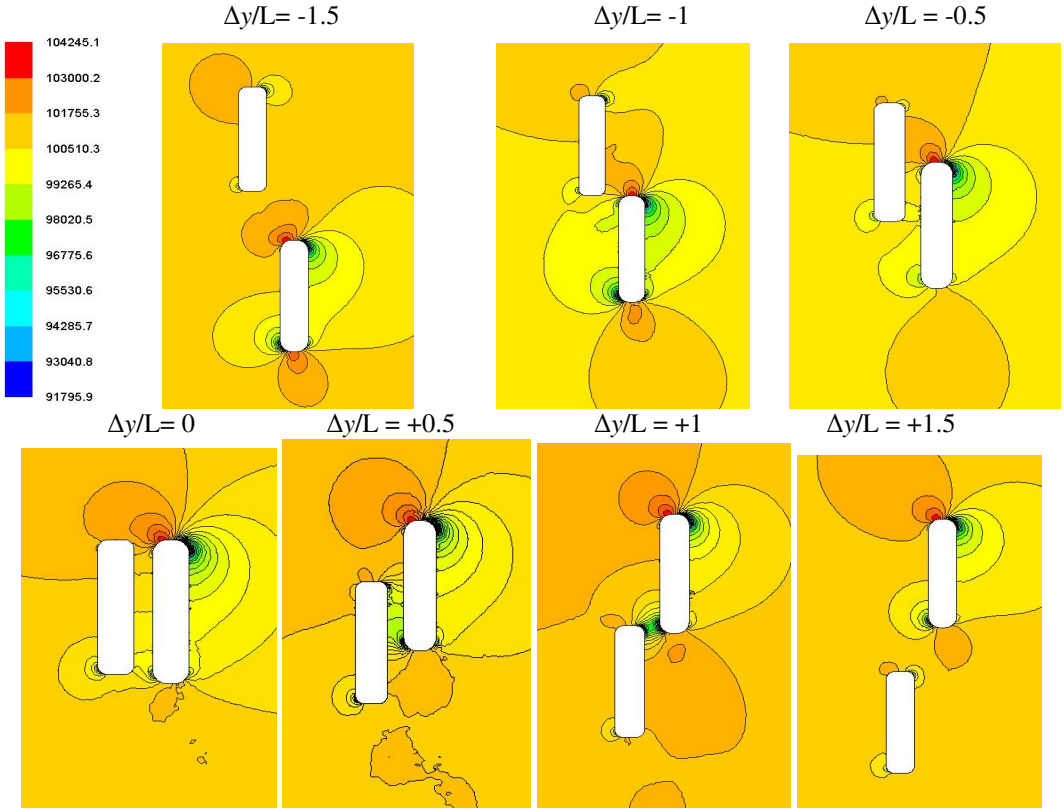


Figure 8
Absolute pressure contours of transient 20º cross flow condition with dynamic mesh

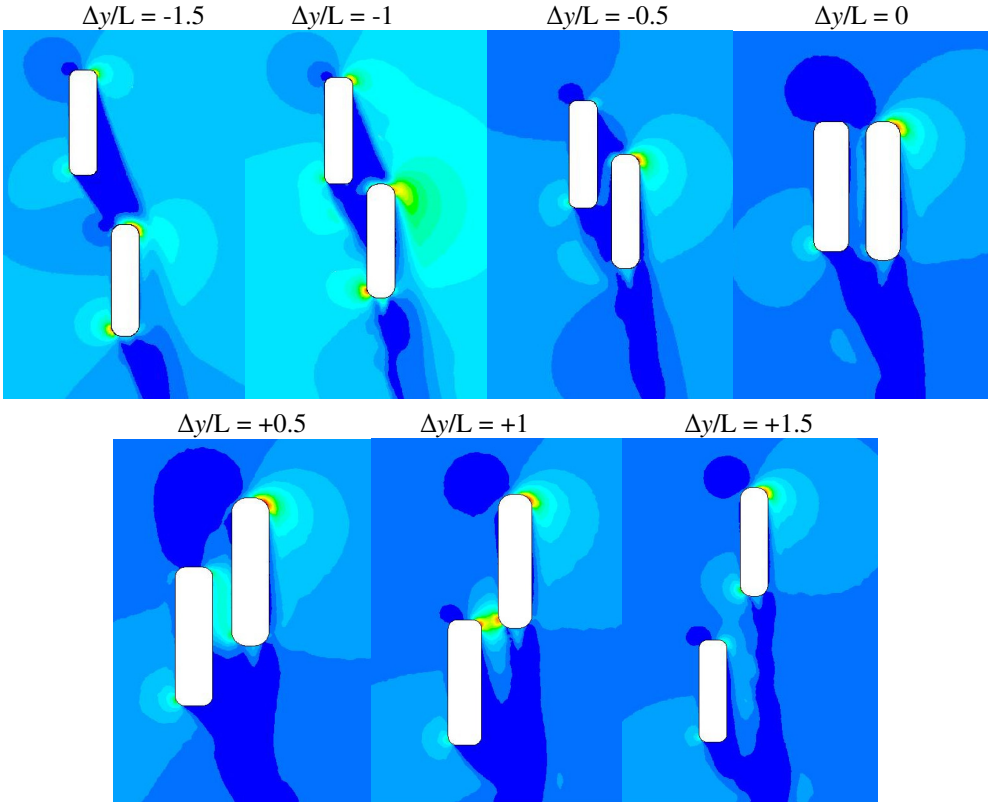


Figure 9
Dynamic pressure variation along length

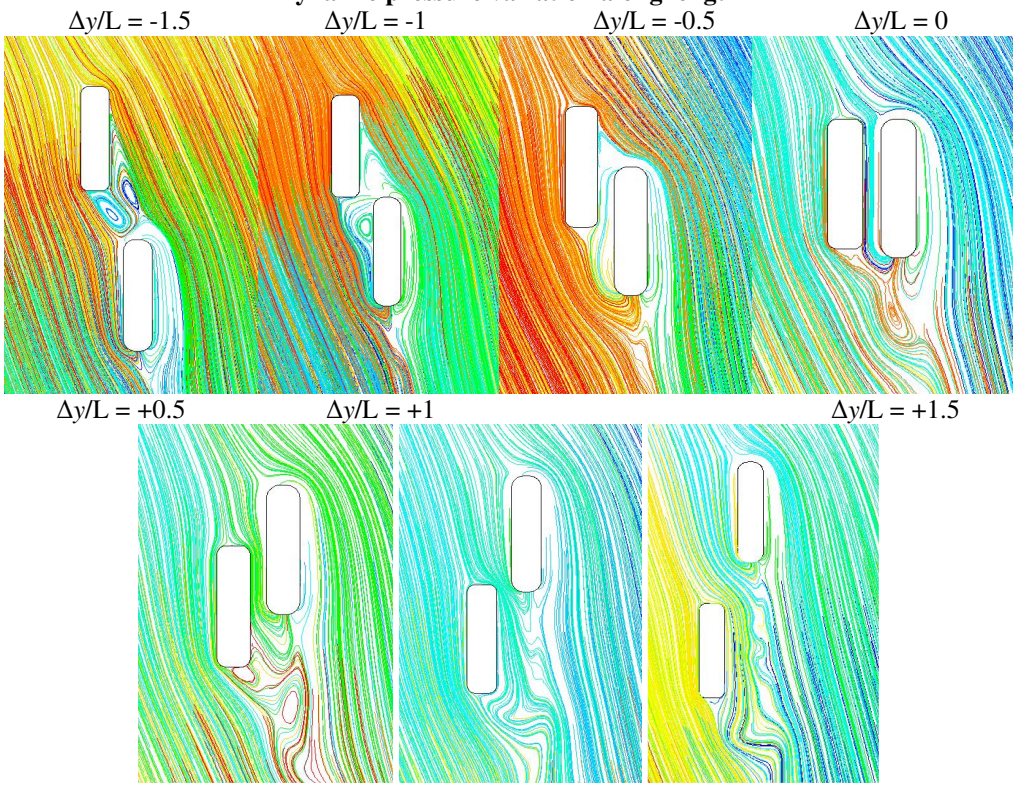
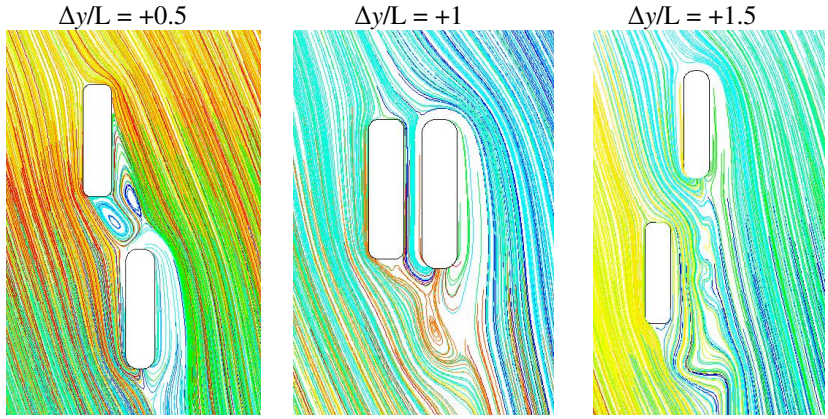


Figure 10
Pathline of transient case with dynamic mesh



a) b) c)

Figure 11
Flow separation, suppression but no re-establishment

Apart from monotonous increment in pressure, flow separation re-establishment is absent in transient case while in fig 6 it is re-established in steady state which clearly reveals that only steady state solution is not enough for overtaking type of problems. At wake region pressure also high pressure is building up due to the main flow energy source which also contributing in the evaluation of constant values. Other parameters should also consider which will contribute for fluctuating the side force and yawing moment.

Conclusion

Transient condition is inevitable for cross flow condition. Stagnation pressure is high at windward side compare to steady case, so more prone to yawing moment and side force which leads to instability of the vehicles. Path line contours depict that there is no flow re-establishment at particular time like in steady state condition which is depicted in Fig 6. While designing the vehicle according aerodynamic way, following parameters must be considered in terms of controlling the yawing, side force for critical condition like cross wind.

Tapered front section will slightly decrease the yawing moment and side force effects on overtaking vehicle. Notchback exhibits highest yawing moment and square back will lowest while reverse for side force. Rounding of the edges of front end and hood and hood sides increase the yawing moment and leads to lower pressure on leeward side of front end. Increase in side projection area will leads to decrease in side force at rear end but will increase in yawing moment due to increment in length. Low placed cowl with less extent and greater wind shield angle reduce the front side force and yawing moment.

Regarding the present work one can simulate actual condition of highways like more than two vehicles with dimensions modification as explained earlier either the phenomenon frequently happening like car is overtaking the truck etc.

Symbols and Notations: C_d = Drag-force co-efficient, C_p *(s)* = Static pressure co-efficient, C_s = Side force co-efficient, C_{ym} = yawing moment co-efficient, D = Drag force (N), M = Yawing moment (Nm), P_s = Static pressure (N/m^2), P_∞ = free stream static pressure (N/m^2), R_e= Reynolds number, SF= Side force, CFD = Computational Fluid Dynamics, V = velocity of overtaken vehicle (vehicle_2) (m/s), V_{res}= Resultant velocity (m/s), V_{cw}= Cross wind velocity (m/s), V_r = relative velocity between two vehicles (m/s), W= Width of overtaken vehicle (m), L= Length of overtaken vehicle (m), Δx= Lateral spacing between two vehicles (m), Δy= Longitudinal spacing between two vehicles (m), ρ = Density of air (kg/m^3)

References

1. Wolf Henrich Hucho, Aerodynamics of Road Vehicles. SAE International, Warrendale, PA **(1998)**
2. Okumura K. and Kuriyama T., Transient aerodynamic simulation in crosswind and passing an automobile, SAE Paper 970404 **(1997)**
3. Gillieron P. and Noger C., Contribution to the analysis of transient aerodynamic effects acting on vehicles SAE Paper 2004-01-1311 **(2004)**
4. Sims-Williams D.B., Self-excited aerodynamic unsteadiness associated with passenger cars, Ph.D. Thesis, University of Durham **(2001)**
5. Corin R.J., He L. and Dominy R.G., 'A CFD investigation into the transient aerodynamic forces on overtaking road vehicle models', *Journal of Wind Engineering and Industrial Aerodynamics*, **96,** 1390–1411 **(2008)**
6. Fluent User Guide 6.3
7. Krajnovic S. and Davidson L., Large-eddy simulation of the flow around a simplified car model, SAE Paper No. 2004-01-0227 **(2004)**

8. John D. and Anderson Jr., Computational Fluid Dynamics-basics with application, McGraw-Hill series in mechanical engineering **(1995)**

9. Kumar Krishan and Aggarwal M.L., A Finite Element Approach for Analysis of a Multi Leaf Spring using CAE Tools, *Research Journal of Recent Sciences*, **1(2),** 92-96 **(2012)**

10. Purkar T., Sanjay and Pathak Sunil, Aspect of Finite Element Analysis Methods for Prediction of Fatigue Crack Growth Rate, *Recent Journal of Research Sciences,* **1(2),** 85 -91 **(2012)**

11. Renganathan Manimaran, Rajagopal Thundil Karuppa Raj and Senthil Kumar K., Numerical Analysis of Direct Injection Diesel Engine Combustion using Extended Coherent Flame 3-Zone Model, *Res. J. Recent Sci*, **1(8),** 1-9 **(2012)**

12. Magarajan U., Thundil Karuppa Raj R. and Elango T., Numerical Study on Heat Transfer of Internal Combustion Engine Cooling by Extended Fins Using CFD, *Res. J. Recent Sci*, **1(6),** 32-37 **(2012)**

13. Atre Pranav C. and Thundil Karuppa Raj R., Numerical Design and Parametric Optimization of Centrifugal Fans with Airfoil Blade Impellers, *Res. J. Recent Sci*, **1(10),** 7-11 **(2012)**

14. Chauhan Rajsinh B. and Thundil Karuppa Raj R., Numerical Investigation of External Flow around the Ahmed Reference Body Using Computational Fluid Dynamics, *Res. J. Recent Sci*, **1(9),** 1-5 **(2012)**

15. Thundil Karuppa Raj R. and Ramsai R., Numerical Study of Fluid Flow and Effect of Inlet Pipe Angle in Catalytic Converter Using CFD, *Res. J. Recent Sci*, **1(7)** 39-44 **(2012)**