Aerodynamic analysis of vehicle using CFD

Nakul Bansod[1] Deepak Choudhary[2]
1,2Assistant Professor,
1,2Department of Mechanical Engineering
S.V.I.T.S. Indore

Abstract—Maximizing fuel efficiency of a vehicle is one of the prime areas of focus in the highly competitive automotive industry which requires development of efficient and optimized vehicle designs. External aero analysis using Computational Fluid Dynamic (CFD) techniques is widely used in the accurate estimation of an automotive vehicle's drag coefficient, often critical in determining the fuel efficiency of the vehicle and thus drives the design development process. The current computational methods in aerodynamics are a natural result of a continuous search for the understanding of fluid dynamics. In this paper we will discuss the use of CFD analysis for three different cases. In first case, CFD analysis of a Heavy Commercial Vehicle is discussed to improve its fuel efficiency and balancing aerodynamic forces over it. In the second case, the stability analysis of a vehicle while overtaking is analysed using CFD and in the final case, CFD approach for analysing the internal flow field and overall ventilation rates inside a livestock carrying containers is presented.

Keywords- Boundary Conditions, Drag, Wind Tunnel

I. INTRODUCTION

The use of Computational Fluid Dynamics (CFD) codes by the engineering community to predict aerodynamic flow around ground vehicles has increased dramatically in the last few years. This rise in interest and use has resulted from improvements in the predictive capabilities of codes, reductions in the cost of computing technology, and inflation of the costs to perform experiments and to maintain experimental facilities. Most industrially relevant geometries are usually defined in the CAD environment and must be translated and cleaned up to generate water-tight surfaces for simulations using the standard body-fitted approach. This process is very tedious and time consuming. In addition, during this process small details are usually eliminated and overlapping surface patches are trimmed. A smooth water-tight surface mesh is then made which serves as a boundary condition for the volume mesh.

CFD Analysis of a heavy commercial vehicle

There are mainly two types of approaches in volume meshing, structured and unstructured meshing. In structured meshing, the governing equations are transformed into the curvilinear coordinate system aligned with the surface. It is trivial for simple shapes, however, it becomes extremely inefficient and time consuming for complex geometries. In the unstructured approach, there is no transformation involved for governing equations.

In the IB method, the surface geometry would still be present but the volume mesh is generated irrespective of the surface mesh. The IB technique is incorporated in the framework of a Reynolds- Averaged Navier-Stokes (RANS) simulation scheme. The resulting analysis tool provides flow field solutions directly from a CAD representation of the geometry. This represents a significant advance in the application of CFD to realistic flows which can help the industry for fast and efficient design iterations. It uses Cartesian-like meshes in a simple, fictitious computational domain obtained by eliminating the object of interest (i.e. the road vehicles in the present application).

Experimental study

Flow measurements: The experiments were carried out in a small wind tunnel. Surface mean pressure measurements are obtained using several pressure taps installed on the symmetry plane and as well as the base of the model. The velocity field was measured using Particle Image Velocimetry (PIV).

Numerical simulation system: The IB-RANS simulation system consists of three components: (1) the pre-processor that handles the geometry modelling and the grid generation; (2) the flow solver and (3) the post-processor that produces the flow maps on the immersed surface. Each component is described briefly in the following sections.

i) Pre-processor: The pre-processor performs three major functions namely the grid generation, the interface and the interior cell determination, and the evaluation of the weighting coefficients for the immersed boundary interpolation. Geometries are imported in Stereo-Lithography (STL) format. The STL representation of a surface is a collection of unconnected triangles of sizes inversely proportional to the local curvature of the original surface.

ii) Flow solver: The IB approach and the treatment of the governing equations in this investigation are described by Kang et al.; and Iaccarino et al. Briefly, a second-order, cell-cantered, finite volume scheme on Cartesian non-uniform grids is used to solve the steady-state incompressible Reynolds-Averaged Navier–Stokes equations. The momentum equation is solved sequentially for each component of an intermediate velocity.

iii) Post processing: In the IB approach the immersed surface is disconnected from the computational grid and therefore it is not trivial to represent surface information. The procedure used is shown schematically. In addition to the surface quantities, the
postprocessor can also be used to get the flow data on any user specified planes. Currently, the postprocessor outputs solutions in Tecplot ascii format, however, utilities are available to convert them to other popular formats.

IV) Local grid refinement (LGR): Cartesian methods have difficulty in dealing with geometries with steep gradients because of the overhead associated with the local resolution (even if fine grids are only required in a limited region, grid lines must be extended to the boundary of the domain). Also in the context of the IB technique, very refined grids are often required close to the high curvature immersed surfaces in order to properly represent the details of the geometry. Here an LGR algorithm is used which allows an efficient clustering of cells for Cartesian meshes in the vicinity of the immersed boundary. The present implementation is an extension of the classical Adaptive Mesh Refinement (AMR) technique for non-isotropic boundary refinements.

Stability analysis of a vehicle while overtaking
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 manoeuvres have traditionally been completed using the quasi-steady approach. The quasi-steady approach assumes that the relative velocity of the vehicles has a negligible effect and the flow conditions remain approximately steady during the overtaking manoeuvre. As such, the overtaking manoeuvre is analysed with the vehicles at a number of discrete static positions. The validity of the quasi-steady approach and the impact of the ratio between the relative velocity and the overtaken vehicle velocity \( k = \frac{1}{4} \frac{V_r}{V} \) is an area of some debate.

Computational simulation approach
The CFD package used for this study was developed by Professor Li He of the University of Durham primarily for the investigation of unsteady stator–rotor interaction in axial flow turbines. The computational solver adopts a cell-centred finite volume approach for which the temporalintegration of the discretised equations is completed using the four-step Runge–Kutta time-marching scheme. A time-consistent multi-grid technique is used to accelerate the unsteady calculations.

1. Computational mesh design
The size of the computational domain for the overtaking simulation was set at 2 L either side of the models and 3 L upstream and downstream of the models. This domain size was consistent with that used by Okumura and Kuriyama (1997) and was deemed adequate to capture the changes in the flow field during the interaction while giving sufficient distance between the models and the boundaries.

2. Boundary conditions
The outer boundaries of the computational domain were linked to the unbounded far field boundary conditions for which stagnation pressure, stagnation temperature and flow speed and direction were set. At the boundaries between fixed blocks the solver determined the flow variables from the corresponding boundary points in the adjacent block using linear interpolation.

3. Turbulence modelling
The one-equation Spalart–Allmaras (SA) model was employed and the recirculating near wake described by Bearman was observed at the rear of the isolated PARAD1 model. For many, bluff body problems the position of separation is governed by the geometry of the body. However, for cases without a clear, geometry-defined separation point the problem is more complicated. While details about the boundary layer thickness and pressure gradients can be obtained from the Reynolds Number and body geometry, the point of separation is strongly dependent on the flow structures and turbulence production within the buffer region of the boundary layer.

CFD approach for analysing the internal flow field
In recent years, increasing public and governmental concerns about the potential threat to animal welfare during transport have prompted research into the factors affecting the micro-climate experienced by animals during transit. A small number of studies have considered the influence of aerodynamics on the micro-climate within large transportation vehicles demonstrating that: (i) the vehicle motion leads to highly turbulent external flow fields with separation occurring around the front of the vehicle, which is instrumental in driving the airflow from back to front, i.e. opposing the free stream and (ii) four major factors directly affect the internal flow: vehicle speed, wind direction, vent area and the degree of blockage due to the animals’ presence, and that such information can be exploited to guide the placement of fans in low-pressure regions to enhance ventilation mechanically.

Wind tunnel experiments
A series of wind tunnel tests was performed using a 17th scale model of a combined towing vehicle and simplified livestock trailer, the latter constructed from Perspex panels to facilitate flow visualisation. The trailer had two separate decks, each with a series of eight rectangular vent apertures positioned on the side panels. In addition, a large rear vent spanning the entire width of the trailer was placed above the tailboard. A small number of local air velocity measurements were made using a simple stagnation tube, with the local static pressure referenced from a rear surface tapping which measures the base pressure in this fully separated region.
Results and discussion
The aerodynamic and ventilation characteristics of the trailer are now investigated, including the effects of introducing animals within the empty trailer and of making changes to the geometry of the towing vehicle. Finally, Reynolds number effects are addressed via a brief analysis of the full-scale problem.

1. **Drag:** Recent substantial increases in fuel prices have brought the need for minimal aerodynamic drag into much sharper focus. For the case considered above, the combined drag coefficient for the scale-model towing vehicle and trailer is 0.714, which equates to a total drag force of 7.66 N. Of this total, approximately 40% is attributable to the trailer.

2. **Internal air velocities and flow structure:** When considering the flow patterns inside the trailer, the velocity field plays a significant role in dictating where the air actually goes. Moving to the upper deck, the air speeds are not substantially higher than those on the lower deck, but larger velocities are present in the rear half of the trailer. These elevated velocities are a result of air from the free-stream entering the rear side vents and passing out through the top of the tailboard. In fact, this passage of air serves to drive the internal flows in the direction of the free-stream with a more clearly defined structure than that seen on the lower deck, Fig. 2(b). Note that this flow regime is quite different from that found for larger, single-unit animal transporters, where air is drawn into the rear of the transporter in the opposite direction to the free-stream.

II. CONCLUSIONS
Computational fluid dynamics (CFD) methods were successfully developed and applied to the study of the aerodynamic forces on overtaking road vehicles. The study involved simplified 2-D representations of road vehicles and focused on the effect of the relative velocity of the vehicles and the influence of a crosswind on the transient aerodynamic forces. The CFD investigation into the impact of relative velocity highlighted the dynamic amplification of the strength of the pressure fields around the vehicles involved in the overtaking manoeuvre. The results confirmed that the quasi-steady approach captures the overall pseudo-periodic trends in the force variation during an overtaking manoeuvre. However, dynamic effects occurring at relative velocities typical of motorway driving conditions prevent the quasi-steady method from accurately predicting the peak magnitudes of these aerodynamics.

III. REFERENCES
[1] R.J. Corin, L. He, R.G. Dominy “A CFD investigation into the transient aerodynamic forces on overtaking road vehicle models” Journal of Wind Engineering and Industrial Aerodynamics, Vol. 96 (2008), pp 1390–1411
[2] Bahram Khalighi, Shailesh Jindal “Aerodynamic flow around a sport utility vehicle Computational and experimental investigation” Journal of Wind Engineering and Industrial Aerodynamics, Vol. 107-108 (2012), pp 140-148
[3] J.C. Dai, Y.P. Hub, “Aerodynamic loads calculation and analysis for large scale wind turbine based on combining BEM modified theory with dynamic stall model” Renewable Energy, Vol.36 (2011), pp 1095-1104
[4] C.A. Gilkeson, H.M. Thompson, “An experimental and computational study of the aerodynamic and passive ventilation characteristics of small livestock trailers” Journal of Wind Engineering and Industrial Aerodynamics, Vol.97 (2009), pp 415–425.