ISSN 2348 – 8034
Impact Factor- 5.070

GLOBAL JOURNAL OF ENGINEERING SCIENCE AND RESEARCHES

AN AERODYNAMIC STUDY OF AIR FLOW OVER A TRAIN TO SPECIFY THE MAXIMUM AIR INLET

Mr. Shubham Zade*[1], Prof. Shridhar Tambade[2], Mr. Yash Gaddamwar[3] & Mr. Amey Kate[4]
[*1,2,3&4]Department of Mechanical Engineering, BVCOE, Lavale, Pune

ABSTRACT

This paper consists of an aerodynamic study of air flow over a train to understand flow pattern and maximum air inlet for another project work. This research work identified a major source of vibration and energy loss associated with flow separation and vortex formation in the gap between the engine and coach next to it in a train. This study gives idea to identify efficient running in the train in terms of fuel consumption, stability of the train and passenger comfort during motion. There are several solutions related to the drag occurring in the train but the most efficient solution to the problem is flexible polymeric sheets may be connected at the gap between engine and the coach. In this review, we also specify the maximum air inlet and flow pattern at the surrounding of the bogie. In this paper CFD analysis is studied thoroughly.

Keywords: *Aerodynamic Study, Flow Separation, CFD analysis.*

I. INTRODUCTION

This paper considers the aerodynamic forces acting on container trains. It provides a reappraisal of some key test results that have been the basis of much analysis. The aerodynamic drag is approximately proportional to the square of (relative) wind speed. An aerodynamic optimization procedure using a genetic algorithm is presented.

This multi-purpose procedure is first applied to drag minimization. In a preliminary phase, the feasibility of the method is assessed on two-dimensional profiles. In addition to specific features of the genetic algorithm, the implementation of automatic mesh generation and flow calculation is investigated. First three-dimensional results are discussed, where the shape generation process is automated.

The moving train shifts the air with it and deforms the moving-through environment. If the train advances with the constant speed on open track, i.e. without presence of other trains or objects to cause interaction with the train being observed, the form of flow is independent of time and the phenomenon are stationary.

When the train moves with non-constant speed or when its immediate environment has been modified by the presence of other passing train or any other obstacle along the railway such as the pedestrian, vehicle, bridge, building, tunnel, etc., the flow of air varies with time and the phenomenon are non-stationary.

Non-stationary aerodynamics studies the effects of the pressure waves occurring when the train moves in the vicinity of single fixed installation, such as wall, bridge or other infrastructure, when passing through the tunnel or passing other train on open track or within the tunnel. In such case, the interaction between the train and its environment shall be determined as well as the safety and comfort limits shall be defined.

II. METHODOLOGY

There are different ways to obtain the velocities and pressures around a moving body. These include full-scale measurements, physical modeling, analytical Solutions and computational fluid dynamics. The analytical solution is possible only for very simplified two dimensional laminar cases, which are not of engineering interest. For many cases the full-scale measurements are difficult and/or expensive to perform. Although physical modeling is feasible for many engineering cases it also suffers from many drawbacks, which include scaling problems and measurement constraints. For instance, measuring the entire three-dimensional velocity field around a moving body is impossible because data can only be obtained from a limited number of points. The alternative is to use the computational fluid dynamics techniques, in which the full three-dimensional velocity and pressure field can be obtained in an affordable way. Fluid flow around moving bodies is normally turbulent, in which the velocity and pressure fields are fluctuating in both time and space. There is no analytical solution for the governing equations of turbulent flows, instead one needs to simplify these equations to be able to solve them numerically. Computational fluid dynamics (CFD) is a technique to solve the governing equations of the fluid flow using complex numerical algorithms.

The equations governing the fluid motion are the Navier-Stokes equations. These equations are derived from first principles of conservation of mass (continuity equation), momentum and energy. The continuity equation takes the form:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u_i)}{\partial x_i} = 0,(1)$$

where $\rho, t, ui$are the density, time and velocity components in the$i$ direction,respectively.
In Equation (1), the density ρ is not constant but depends on the pressure andtemperature.
The momentum equations take the following tensor form:

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(u_j \rho u_i)}{\partial x_j} + \frac{\partial \rho}{\partial x_i} - \frac{\partial \tau_{ij}}{\partial x_j} = 0,(2)$$

Here$\tau_{ij}$is the shear stress tensor.
The conservation of energy yields the following differential equation:

$$\frac{\partial(\rho E)}{\partial t} + \frac{\partial(u_j \rho E)}{\partial x_j} - \frac{\partial}{\partial x_i}\left(k \frac{\partial T}{\partial x_i}\right) + \frac{\partial(u_j p)}{\partial x_j} + \frac{\partial(\tau_{ij} u_j)}{\partial x_j} = 0, \qquad (3)$$

where T is the temperature,τ_{ij} is the shear stress and E is the total energy.

In a turbulent flow, the variables fluctuate randomly in both time and space and statistical approaches can be used to describe the flow. The flow variables can be decomposed into mean and fluctuating a component, which is the basis of the Reynolds decomposition, Flow variables in the present work can be expressed as:

$$u = \bar{u} + u^{'} \qquad (4)$$
$$p = \bar{p} + p^{'} \qquad (5)$$
$$T = \bar{T} + T^{'} \qquad (6)$$

In many cases it is easier to analyze the time-averaged quantities. Thus the Reynolds decomposition is applied into the governing equations to obtain what is called the Reynolds Averaged Navier Stokes (RANS) equations. These equations take the form:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho \overline{u_\iota})}{\partial x_i} = 0,(7)$$

$$\frac{\partial(\rho \overline{u_\iota})}{\partial t} + \rho \overline{u_J} \frac{\partial \overline{u_\iota}}{\partial x_j} = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\rho v \frac{\partial \overline{u_\iota}}{\partial x_j} - \rho \overline{u^{'}_\iota u^{'}_J}\right), and \qquad (8)$$

$$\frac{\partial \bar{T}}{\partial t} + \frac{\partial \bar{T} \overline{u_J}}{\partial x_j} = \frac{\partial}{\partial x_j}\left[\frac{v}{Pr} \frac{\partial \bar{T}}{\partial x_j} - \left(\overline{T u_J} - \bar{T} \overline{u_J}\right)\right](9)$$

for the continuity, momentum and energy equations, respectively. The last term in the momentum equations, $\rho uiuj$are the Reynolds stresses, which are unknown andneed modeling.

The model used to approximate the Reynolds stresses in the present work was the Shear Stress Transport *k*-*ω* (SST *k*-*ω*) model—for more information about the model please see. The commercial CFD package Ansys-CFX has been used together with the sliding mesh technique to solve for the continuity and momentum equation for compressible flow.

III. PROBLEM DEFINITION

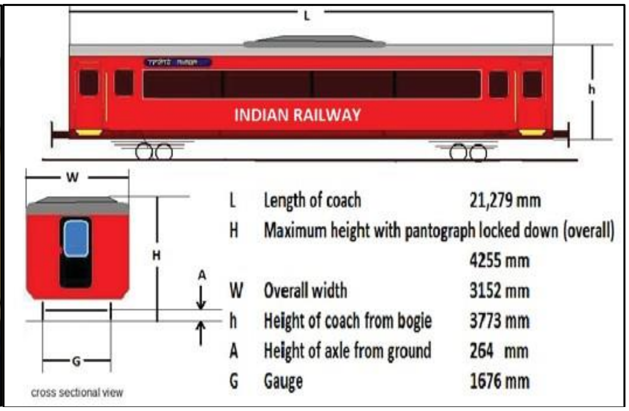
Railcar aerodynamic studies are typically undertaken to improve safety and increase fuel efficiency. As the small modification can gives the change on large scale due to its huge network. So the aim of study is to reduce the aerodynamic drag in the inter-car gap of coaches of train. For this analysis is done on cavity or gap and on a flat plate used as filler between two coaches. This flat plate or filler will reduce the aerodynamic loss on trains. By reducing the drag on coaches, acceleration will automatically increase due which to less power of locomotive require.

IV. GEOMETRICAL SETUP

The analysis must be an accurate model of a physical prototype in dimension and size. In the broadcast sense, this model should be same in the manners of nodes, elements, material properties, real constants, boundary conditions, and other features that can be used to represent the actual system. The object here is referred to the Indian Railways bogies. Dimension and figures of coach is drawn on the bases of official data of Indian railway. IR coaches' whether it is ICF coach or LHB coach faces maximum aerodynamic drag due to their design. Length over body of link Hofmann Busch coach (LHB) is 21,279 mm; width and height of body are 3,125 mm and 3773 mm respectively. Ground clearance of train is 264 mm. Some of the illustrations which display the dimensions are given in Figure 2.

Main motive of the paper is to analyze the flow in inter-car gap and to reduce the aerodynamic drag possessed by the gap. The flow in the gap region is a trapped vortex that does not interact substantially with the free- stream. Adjacent cars with narrow inter-car gaps thus act as a single body. As gap distances increase, the drag approaches that of multiple, single bodies. The generated vortex (eddy) consumes energy of locomotive.

Figure 1: Inter-car gap in Indiantrain



L	Length of coach	21,279 mm
H	Maximum height with pantograph locked down (overall)	4255 mm
W	Overall width	3152 mm
h	Height of coach from bogie	3773 mm
A	Height of axle from ground	264 mm
G	Gauge	1676 mm

Figure 2: front and side view of Indian train with geometrical dimensions

V. COMPUTATIONALSETUP

The ultimate purpose of a finite element analysis is to recreate arithmetically the behavior of an actual engineering system. Flow in inter-car gap behaves as open cavity which study has been carried out with computational domain shown in figure below. This study has three components

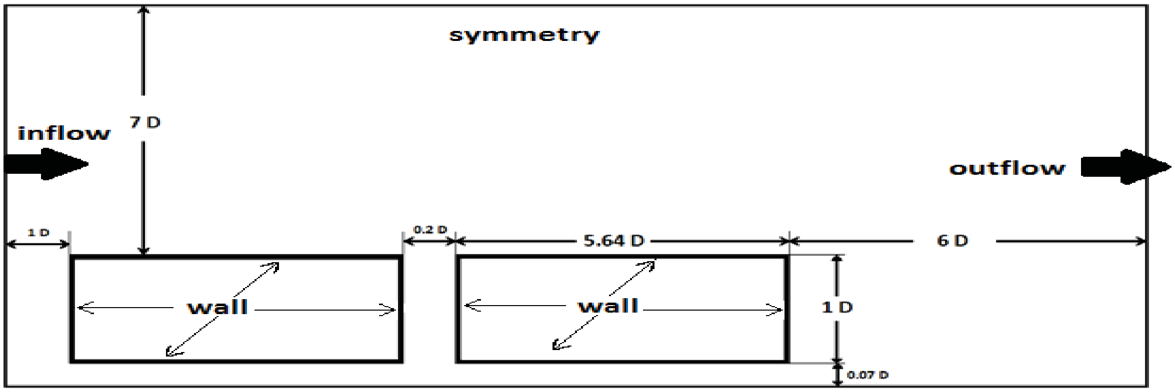


Figure 3: computational dimension of setup

A. **Flow in inter-car gap -** Figure 3 shows the computational dimension of setup. For the analysis the geometry is drawn in 2 D plane. A domain is created in which the railway coaches are carried. The size of domain is 8.07 D vertically and 18.48 D on horizon. Distance of domain from front face of first coach is 1D and at rear it is 6 D. Length of coach is taken as 5.64 D computationally and height is 1 D. Dimension of inter-car gap is 0.2 D; ground clearance of train is 0.07 D. Inlet boundary is nearer to the wall of coach whereas outlet boundary is far from the wall ofcoach.

B. **Flow over the plate of gap -** Figure 4 shows the computational model of coaches with flat plate. Here a flat palateisusedtofilltheintercargap.Thesizeofthisplateissameasthesizeoftheinter-cargap.

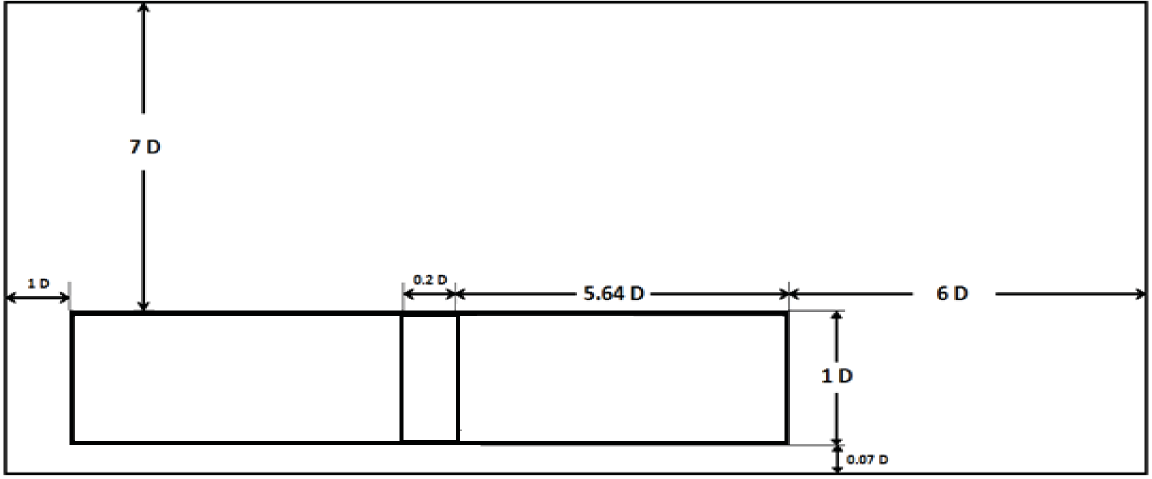


Figure 3: Computational model of coaches with flat plate

C. **Grid generation -** The generation of 2 - dimensional computational model and its meshing has been achieved by a pre-processor named GAMBIT 2.2.30. For both the cases of with and without flat plate, all the faces have been meshed with quad mesh. The domain is divided into number of areas so that each area can be meshed with quad mapping scheme. Fine grids are generated near the walls or where the flow dynamics is complex such as inter car gap where the vortex are generated. The fine grids are placed in such a manner so that the vortex generation in cavity wall y+<1 criterion can be fulfilled. Meshing in case of without cavity is fined near to the upper and lower walls of the coaches andplate.

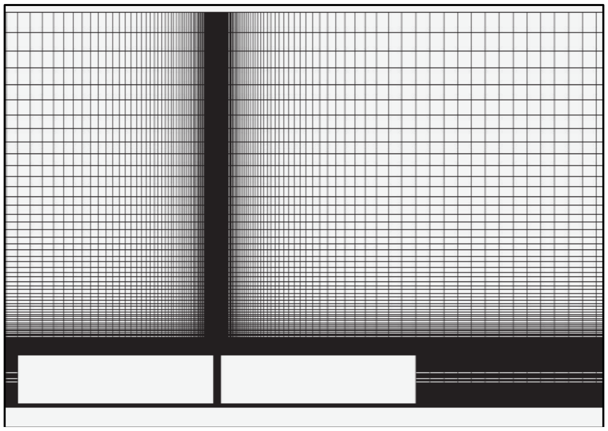


Figure 5: Grid view of full model

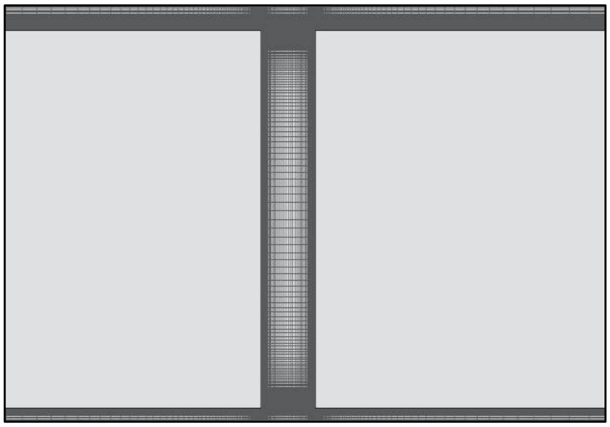


Figure 6: Grid view of inter-car gap area

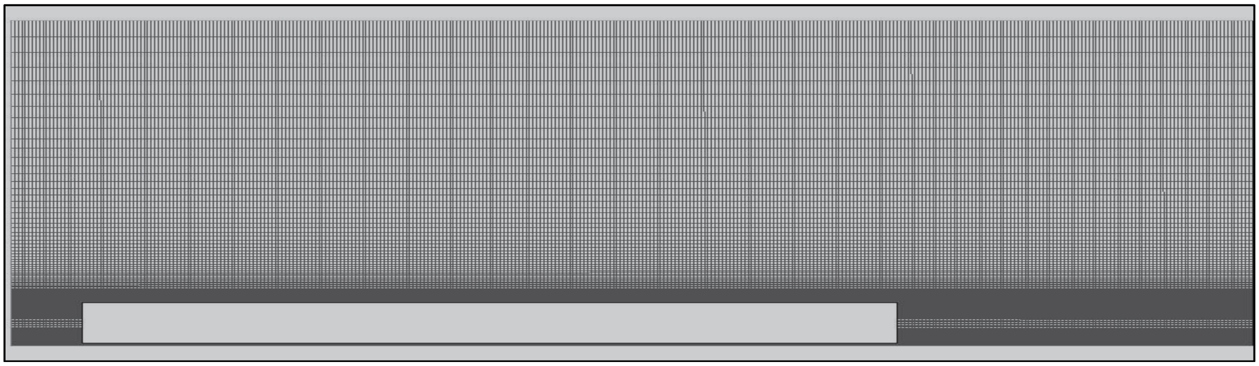


Figure 7: Grid view of model when gap is not available

VI. CONCLUSION

In the study, the aerodynamic drag reduces computationally in the inter-car gap of trains by using the flat plate over the portion of cavity. The size of inter-car gap in the train is 0.75 meter. The study is carried for the ambient condition in which the velocity of train is 20 m/s, due to which the Reynolds number increases, higher Reynolds number is responsible for the turbulence. This turbulence is defined by the k-s standard turbulence model, for this model intensity and length scale has been discussed above.

By studying this we come to the conclusion that the maximum air inlet in a train is on the over head or roof portion. So, the ideal space to mount the intake converging nozzle is at top side of the train. Secondary space to mount the intake converging nozzle is between the space between to wagon i.e. 0.75m.

A detailed flow has been presented with and without the gap in train. According to the results, in the gaped area the wake flow generates due to which vortex is formed, this can be reduced by filling the gap by some plate or by air bags. The result with continued train is good which shows the reduction of aerodynamic drag over the walls of train. Analysis and mathematical calculation shows that, by using the filler in the gap the aerodynamic loss reduces up to 3- 4 % for a gap between two coaches. If a locomotive having 20 bogies moving with the higher speed for a long journey then the aerodynamic loss can be reduced on very big scale.

REFERENCES

1. *https://en.wikipedia.org/wiki/Swachh_Bharat_mission*
2. *http://www.indianrailways.gov.in/railwayboard/uploads/directorate/finance_budget/Budget_2015-16/White_Paper-_English.pdf*
3. *F. Monnoyer, E. Lorriaux, N. Bourabaa, Aerodynamic optimization of railway motor coaches*
4. *W. Cherdron, F. Durst, J.H. Whitelaw, Asymmetric flows and instabilities in symmetric ducts with sudden-expansion, J. Fluid Mech., 84, pp. 13–31 1978.*
5. *I. J. Sobey, P.G. Drazin, Bifurcation of two-dimensional channel flows, J. Fluid Mech., 171, pp. 256–287, 1986.*
6. *Chris Baker, birminghamcentre for railway research and education school of civil engineering gisbertkapp building, university of birmingham, edgbaston, birmingham, United Kingdom, b15 2tt*
7. *F. Durst, J.C. Pereira, C. Tropea, The plane symmetric sudden-expansion flow at low Reynolds numbers, J. Fluid Mech., 248, pp. 567–581, 1993.*
8. *Mirjana Puharic, Marina KutinAerodynamic research of high speed trains in the subsonic wind tunnelTome V (year 2007), Fascicole 2, (ISSN 1584 – 2665)*
9. *T. Hawa, Z. Rusak, Viscous flow in a slightly asymmetric channel with a sudden expansion, Phys. Fluids, 12(9), pp. 2257–2267, 2000. [Design and Fabrication of Garbage controlling Equipment integrated in Train] B.V.C.O.E.L. Pune. B.E (Mechanical) 2018-19 Page 25*
10. *T. Hawa, Z. Rusak, The dynamics of a laminar flow in a symmetric channel with a sudden expansion, J. Fluid Mech., 436, pp. 283–320, 2001*