International Journal of Advanced Mechanical Engineering.
ISSN 2250-3234 Volume 4, Number 5 (2014), pp. 463-468

http://www.ripublication.com

Study of Flow Structure and its effect on Indian Train Using C.F.D. Technique

Vivek Kumar and Abhishek Pratap Singh

Student, B.Tech. Mechanical, ITM University, Gurgaon, Haryana
Deptt of Mechanical Engg., ITM University, Gurgaon, Haryana

Abstract

Present research aims to Study of the Flow Structure and its effect on Indian Trains using C.F.D. Technique. The purpose of the study was to simulate and analyze the flow structure and finding the high pressure points and suggesting suitable changes to eliminate them by low pressure points and study the velocity pattern of air across the body. This analysis is done on 2-D Structure of the concerned object. Objective of this research is to study the air flow over the Indian Railways and to find out the high pressure points created due to the motion of the vehicle. In order to achieve the desired result, C.F.D technique is used and software on which project work is carried out is ANSYS Fluent. Results show that the current design of the locomotive and the bogie has certain flaws. If these flaws are removed, design of the vehicle will be more attractive in looking, it will be more aerodynamically stable and speed and efficiency of the system will also get increased. Centre of the gravity of the vehicle will also be lowered as the ground clearance can be reduced. Current design faces unwanted amount of drag force and more material consumption in manufacturing. High pressure points are generated on the front of the locomotive and the engine has to do extra work to overcome the drag so created by these points.

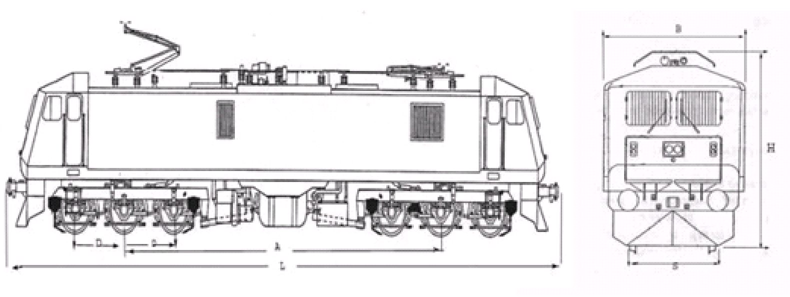
Keywords: Indian Train, Locomotives, Bogies, Air, Fluid flow, C.F.D., ANSYS, Fluent.

Introduction

Certain Problems which are identified when we compare Indian Railways with the railways of some other countries. In India, average number of wagons are around 23 for long distance travel whereas in countries like Australia number goes up to 48. The fastest train in India is Bhopal Shatabdi between New Delhi – Bhopal Junction with a maximum speed of 161 km/h (100 mph) and an average speed of 87.17 km/h (56.5 mph), excluding stops. In India, design of the engine is "old school" i.e. No new development is made in a long time, only old designs are modified a bit, but no new concepts are made whereas in other countries proper R&D departments are there for research work. Indian trains face max, air drag force as compared to engines of trains in other countries because of their old design. Indian railways also lag in terms of aerodynamics and air flow analysis. Present Study has been done to understand the concepts of fluid dynamics and its application, and to study, the airflow over the Indian Railways, behavior of air at different locations, and aerodynamic concepts involved. The study identified the various high pressure points created due to the motion of the vehicle, velocity variation of air when it comes in contact with the vehicle. Present study suggested changes in the design and shape of the engine for better working and efficiency, improvement of the shape and design of the engine, improvement in the looks of the train, and, proper air flow over the engine, hence less effect of air drag force, and more speed of train for the same amount of fuel.

Geometrical Model Setup:

The ultimate purpose of a finite element analysis is to recreate mathematically the behavior of an actual engineering system. In other words, the analysis must be an accurate mathematical model of a physical prototype. In the broadcast sense, this model comprises all the nodes, elements, material properties, real constants, boundary conditions, and other features that are used to represent the physical system. The object here is referred to the Indian Railways Locomotive and the bogies. There dimensions and figured is drawn on the basis of the information obtained from the Internet and the official website of Indian Railways. Some of the illustrations which displays the dimensions are given in Figure 1. The process for generating a mesh of nodes and elements consists of three general steps (i) Set the element attributes, (ii) Set mesh controls. ANSYS offers a large number of mesh controls from which we can choose as needs dictate. (iii) Meshing the model.



A	Distance between bogies (center – center)	12000 mm
L	Length over buffers	20562 mm
D	Distance between axles of bogie	1850 mm
B	Overall width	3152 mm
H	Max. height with pantograph locked down(Overall)	4255 mm
S	Gauge	1676 mm

Figure 1. Dimension of the locomotive

Experimental Setup:

The discretized conservation equations are solved iteratively until convergence. Pressure based solver is applicable for a wide range of flow regimes from low speed incompressible flow to high speed compressible flow. We define the input velocity in Absolute Reference Frame and its value in X-direction is 20m/s. Turbulence intensity is set to 1% and turbulent viscosity ratio to 10. In similar fashion, we determine the pressure outlet parameter. Make sure that symmetry parameter has symmetry type selected and the wall parameter has the wall type selected. Next we make sure that the reference values are computed from the inlet.

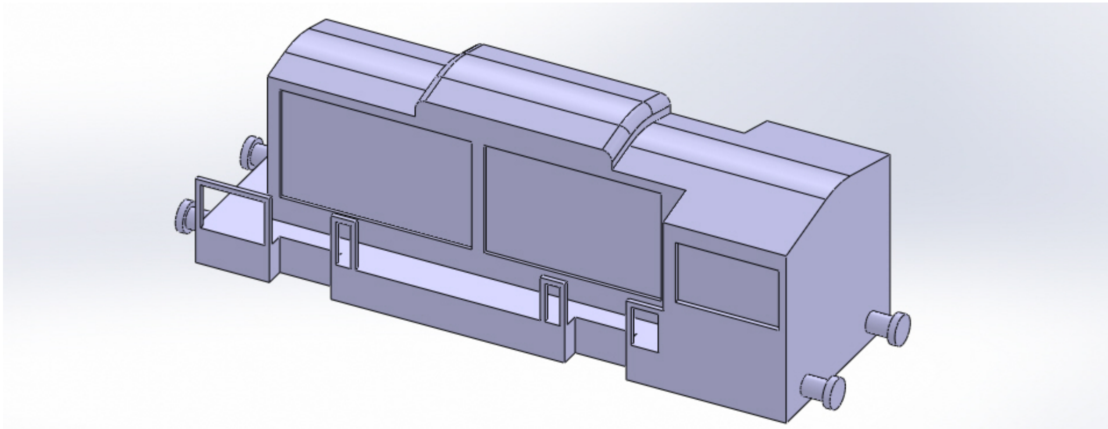


Figure 2. 3-D graphical Model of a Locomotive

Result Discussion

As the flow analysis results are obtained for the single locomotive in motion, it is observed that High Pressure points are developed at the front side of locomotive and under the locomotive. When it comes in term of the velocity, it appears that velocity of air films which are in contact with the locomotive don't vary much. But change in velocity is observed above the locomotive. Low velocity zone is obtained right behind the locomotive which exists till certain distance. Maximum velocity contour is obtained at the front top of the locomotive. The same behavior is depicted by the velocity vectors. Fair amount of velocity contour is obtained below the locomotive. Turbulence is produced over the top of the locomotive and bottom end of the locomotive. As compared to normal velocity of nearby air, turbulence produced is not much in magnitude.

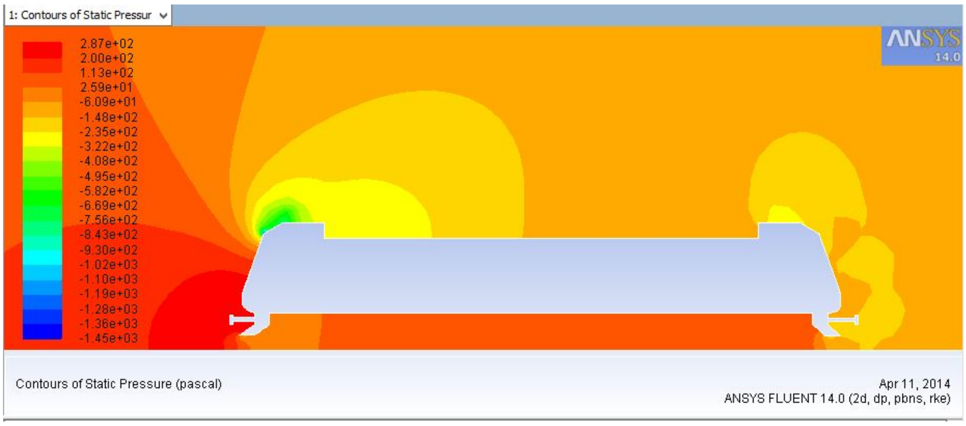


Figure 3. Contours of static pressure on a 2-D locomotive

As the flow analysis results are obtained for the single locomotive and single bogie arrangement, we observed that High Pressure region is formed at the bottom half of the front of the locomotive and a little bit less pressure region is formed at the bottom of the locomotive. On the top of the locomotive, there exists a high pressure region whose magnitude is little less than what's developed at the front of the locomotive. Normal pressure zone is formed above the bogie. After a little back from the bogie, pressure zone starts to become normal again. Turbulence is formed at the top as well as the bottom of the locomotive and large section at the back of the bogie. As compared to surrounding air, its magnitude is not that much. Slight turbulence is also observed at the top of the bogie. Majorly, the High Velocity regions are formed above the locomotive and the bogie. Low velocity regions exist for the air layers which are in contact with body. Low velocity region also exist in the gap between the locomotive and the bogie. Partial low velocity regions are obtained behind the bogie. Velocity region formed below the bogie is more than that is formed below the locomotive.

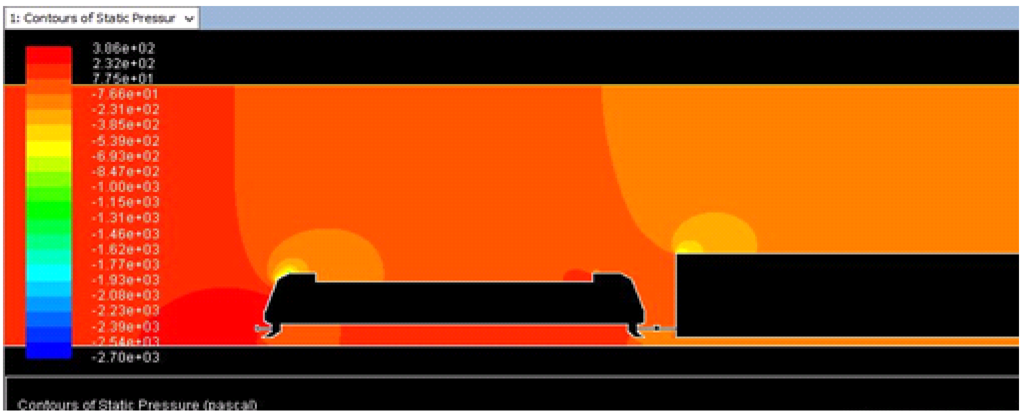


Figure 4. Contours of static pressure on a locomotive and a bogie

As the flow analysis results are obtained for one locomotive and four bogies arrangement, we observed that High Pressure region is formed at the bottom half of the front of the locomotive and at the bottom of the locomotive. A little less pressure region is formed at the top of the locomotive. Around the bogies pressure region is almost same i.e. medium pressure region. Medium pressure regions are also formed behind the bogies. Turbulence is obtained at the top of the locomotive and the top of the first locomotive. A large turbulence region is obtained at the back of the bogies. A fairly good velocity region is observed above the locomotive and the bogies. This region exists till a little distance beyond the last bogie. Very low velocity regions are observed between the locomotive and bogie as well as between the bogies. Small packets of low velocity regions are formed behind the last bogie.

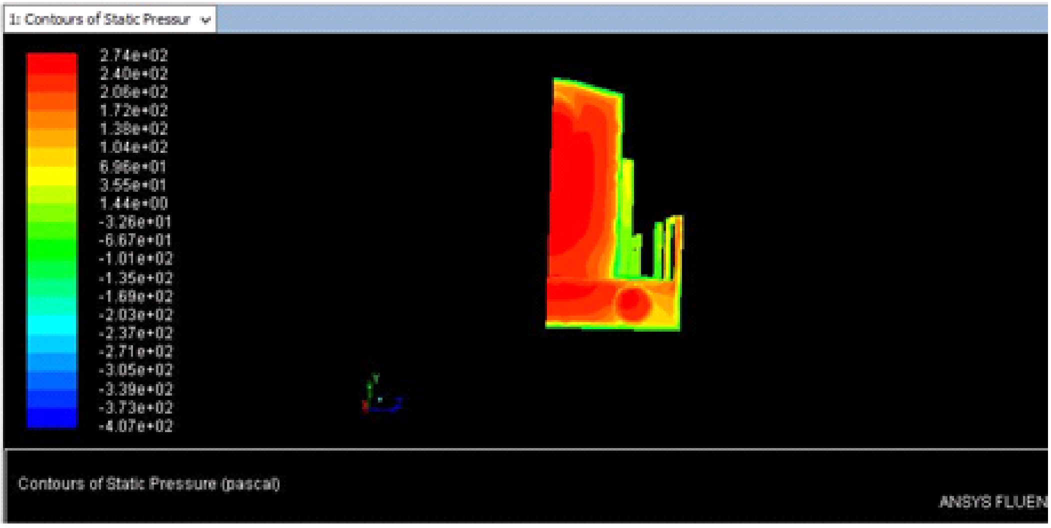


Figure 5. Contours of static pressure on front of a locomotive

Conclusion

It can be suggested that some changes in the design and structure of the locomotive and the bogies which can improve the various things like the speed, aerodynamics, engine efficiency. Changes the shape of the locomotive and make it more pointed out in the front and increase the slope of the front surface then airflow patter will change. Air will flow over the surface to the sideways and resulting in less air drag. It increases the speed of the locomotive and makes the design more aerodynamically stable. Some pressure zone and velocity variations are observed below the locomotive. If we decrease the ground clearance up to some suitable extent then such points can be reduced for good. There are some low velocity regions between the bogies. We can make the bogies in the continuous formation which has same height and cross section throughout the length. In this way a uniform space will be available for continuous airflow.

The present study is aimed at enhancing our understanding of the stated problem and technical capabilities involved in finding a workable solution for the same. In addition, an attempt can be made to find out the extent of application of CFD code FLUENT in the air flow analysis on Indian Railway Locomotive. Further effect of wind orientations, varying aspect ratios, varying locomotive geometries as well varying environmental conditions could be analyzed in the future for more responsive solutions. An attempt could also be made for analyzing Indian Train as a whole i.e. Locomotive and Bogies systematically attached like in real life condition.

References

[1] Tim Langlais, *Ansys Short Course lecture* (langlais@me.um.edu), *on August 16, 1999.*

[2] Dr. Mohammad O. Hamdan, lecture on Ansys Fluent in Spring 2010.

[3] ©Ansys Fluent Theory by Ansys Inc.

[4] Rajesh Bhaskaran, Lecture notes on *Introduction to CFD Basics*

[5] Andre Bakker, *Applied Computational Fluid Dynamics, http://www.bakker.org, ©Bakker(2002-2006).*

[6] Elizabeth M. Marshall and Andre Bakker, Fluent Inc., Technical Notes on *Computational Fluid Mixing, submitted to the North American Mixing Forum (NAMF) Handbook on Mixing January,2001.*

[7] Tao Xing and Fred Stern, Simulation of Turbulent Flow over Ahmed Body, Intermediate Mechanics of Solid, CFD Lab, By:, IIHR- Hydroscience and Engineering, The University of Iowa, C. Maxwell Stanley Hydraulics Laboratory, Iowa City, IA 52242-1585.

[8] Tennekes H., Lumley J.L. "*A First Course in Turbulence* ." The MIT Press 1972.

[9] Versteeg H.K., Malalasekera W. (1995) *"An Introduction to Computational Fluid Dynamics: The Finite Volume Method" by Longman Scientific & Technical. ISBN: 0-582-21844-5.*

[10] Pope, Stephen B. "Turbulent Flows." Cambridge University Press 2000.