Available Online through www.ijptonline.com

FLOW ANALYSIS EFFECT ON N700 SHINKANSEN BULLET TRAIN USING FLUENT SOFTWARE

T. Narendiranath Babu*

1School of Mechanical Engineering, VIT University, Vellore.

Received on 15-05-2016 Accepted on 18-06-2016

Abstract

In this paper, an efficient numerical model for the dynamic interaction analysis of a N700 Shinkansen train (bullet train) and surrounding environment is considered. The purpose of the study was to analyze and simulate 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 a 2-D structure of the concerned object. The motion of a Shinkansen train is modeled in multibody dynamics. A simple and efficient mechanical model for interaction between train and surrounding environment is described. The railway structure is modeled with various finite elements. A software program Ansys-Fluent has been used for the interaction analysis of a Shinkansen train and the surrounding environment.

Keywords: Computational fluid dynamics, Train model, velocity pattern, fluent software.

1. Introduction

The study of the aerodynamics of a train can lead to substantial cost savings and more environmental friendly trains. The most studied phenomenon in the realization of a train is the drag generated by the displacement of the train in the air flow[1-6]. Among the different ways to travel such as airplane, car, bus or boat, the train is the safest. According to the report of the ECST [7], between 2001/2002 the death per 100 million person kilometers is in the European Union 0.035 and the death per 100 million person hours is 2 which makes the rail the safest way to travel before ferry (respectively 0.25 and 8), aircraft (respectively 0.035 and 16) and bus (respectively 0.07 and 2). In addition, with the progress of technology, train is able to drive at high-speeds. On, the 3rd of April 2007 a TGV in France reached the velocity of 574.8 km/h. However this speed is only a record and the TGV trains drive generally between 200 and 330 km/h. The International Union of Railways (UIC) defines that the speed of a high-speed train must be at least 200 km/h for upgraded
track and 250 km/h for new track. For example, the TGV line between Paris and Strasbourg runs between 310 and 320 km/h. When a train runs, a strong head pressure pulse is created at the very front of the train which leads to a change of pressure in the surrounding [8-9]. A low pressure bubble is also created at the rear part of the train but is less strong than the first one, and is mostly a problem for people or objects standing near to the track. With the increase of the train speed, aerodynamics has become an important key in the rail vehicles field. Reducing the drag leads to a reduction of the amount of energy needed. But some others aerodynamic phenomena are of interests, such as the pressure variations while the train is driving in a tunnel, the study of the consequences of a crosswind, the study of the train rollover or the slipstream[10-11]. The pantograph is situated in an area where the flow conditions change a lot. In order to avoid unauthorized large variations of the contact surface it is important that the flow around the pantograph is not too turbulent, which can be enabled by adding some so called fairings. While entering a tunnel the air at the train nose is compressed this creates an overpressure wave that migrates at the speed of sound. When this waves reaches the end of the tunnel a part of the wave is reflected and goes back as an under pressure wave. As the train tail enters the tunnel an under pressure wave is created and migrates to the end of the tunnel. The pressure variation is maximum when an under pressure wave meets a reflected overpressure wave. The pressure difference reaches then a peak value. This mostly causes discomfort for the passenger since he is subject to a high pressure difference in a short time, for example the legislation in Sweden is 1500 Pa in 4 seconds. The bogies movement is restricted to the tracks. A suspension system connects the train body to the bogies. The train can then roll yaw and pitch. A yawing moment can increase in strong crosswinds. This can be very dangerous in case of strong crosswind and particular yaw angle and can lead to overturning. Figure 1 shows model of bullet train

![Figure 1 Model of bullet train.](image)
2. Methodology

The train was modeled using the dimensions:

| Car length                  | 25,000 mm (82 ft 0 in) (intermediate cars)  
|                            | 27,350 mm (89 ft 9 in) (end cars)       |
| Width                       | 3,360 mm (11 ft 0 in)                  |
| Height                      | 3,600 mm (11 ft 10 in), 3,500 mm (11 ft 6 in) (end cars) |

Mesh was generated using the triangles method. Relevance centre was set to fine. Smoothing was set to high. This was done in order to obtain accurate results. Moreover, a refinement method was setup of magnitude 3 at the train wall in order to obtain even greater accuracy. Gravity was taken into consideration. The viscous model taken into consideration was k-epsilon model.

K-epsilon (k-ε) turbulence model is the most common model used in Computational Fluid Dynamics (CFD) to simulate mean flow characteristics for turbulent flow conditions. It is a two equation model which gives a general description of turbulence by means of two transport equations (PDEs). The original impetus for the K-epsilon model was to improve the mixing-length model, as well as to find an alternative to algebraically prescribing turbulent length scales in moderate to high complexity flows.

- The first transported variable determines the energy in the turbulence and is called turbulent kinetic energy (k).
- The second transported variable is the turbulent dissipation (ε) which determines the rate of dissipation of the turbulent kinetic energy.

The material considered was air. The standard values of density and viscosity of air were taken into consideration which was included in the software.

The flow characteristics were obtained at three different velocities

1) 27.78 m/s (100 kmph)  
2) 55.56 m/s (200 kmph)  
3) 83.34 m/s (300 kmph)  

Turbulence Intensity : 100 % (Re > 100000)

Turbulence Viscosity ratio: \( I = 0.16Re^{-1/8} \)
The values were calculated using the above formulae.

Following solution methods were incorporated in the model:

- Pressure Velocity Coupling Scheme: Simple
- Special Discretization:
  1) Gradient – Least squares cell based
  2) Pressure – Second Order
  3) Momentum – Second Order upwind
  4) Turbulent Kinetic Energy – First order upwind
  5) Turbulent Dissipation Rate – First order upwind

500 iterations were provided. Solution was converged at the 64th iteration itself.

The corresponding velocity and pressure contours were obtained and the velocity vectors were plotted as well.

3. Results and Discussion:

3.1 Geometrical Model

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 N700 Shinkansen Locomotive and the bogies. 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.

![Figure 2 Solid model of bullet train.](image)
4. Flow analysis

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. The input velocity is in absolute reference frame. Turbulence intensity is set to 100%. 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.

![Flow analysis of train at 1/3 of velocity](image)

Figure 3 Flow analysis of train at 1/3 of velocity
The different parts of slipstream can be visualized in figures 3, 4 and 5 the pressure pulse at the very front which characterizes the flow around a train which was region 1, the boundary layer which increases along the train which was region 2, the two counter-rotating when the flow separates at the rear part of the train which was region 3 and the disturbed flow in the far wake which was region 4.

It can be observed that in the velocity profile is not symmetrical which is due due to the presence of the platform. The turbulent flow is also visible to characteristic chaotic phenomena. It is worth mentioning that additional velocity plots have been extracted during the course of this work in order to see that there was no strange behavior in the solution. Some parts of the train, especially the inter-car gaps, were found to cause some problems, since some spurious results were visible on velocity plots. To conclude, this part of the model requires very fine and precise grid in order to have accurate results.

The aim of the paper was to study the slipstream which is an aerodynamic phenomenon caused by the viscous air which is dragged when a train is passing and which can destabilize people or objects on a platform near the train. To avoid this problem, the train must fulfill some requirement such as the Technical Specification of Interoperability.

The flow has been computed for different Reynolds numbers by changing the size and/or the velocity in order to see the difference for the drag and lift coefficients depending on this number.

The train surfaces are affected by strong shear stress. The area where the friction force significantly affects the air velocity is called the boundary layer. Inside it, the flow can be either laminar or turbulent. The length of the train being quite long, the flow inside the boundary layer is assumed turbulent which leads to strong local velocity variation.
Figure 4 Flow analysis of train at 2/3 of velocity.
5. Conclusion

From the contour plots obtained from Fluent, we can observe that high pressure points are observed near the sharp edges. This may be attributed to the nature of Turbulent Flow. Whenever a high speed flow encounters a sharp edge, three dimensional vortices are formed whose energy is eventually dissipated by energy cascade i.e transfer of momentum and energy from the larger eddies to the smaller ones.

Hence, this technique is used to eliminate these high pressure points so that occurrence of a failure is minimized. Also, from the graphs above, it can note that an increase in velocity of the train results in a great increase in the pressure at various points of the engine. Hence, an aerodynamic shape has been adopted by the bullet trains to obtain minimum pressures. A study by Tai et al. shows that the golf ball effect that reduces drag on dimpled golf balls also works on a car. Hence, if a similar concept is applied to trains, overall drag and fuel consumption would be reduced that would result in improved efficiency.

References:


Corresponding Author:
T. Narendiranath Babu*

Email: narendiranathbabu.t@vit.ac.in