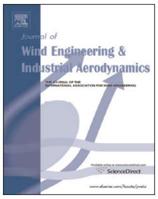
J. Wind Eng. Ind. Aerodyn. 145 (2015) 87–93

Contents lists available at ScienceDirect

Journal of Wind Engineering and Industrial Aerodynamics

journal homepage: www.elsevier.com/locate/jweia



Numerical study of the impact of windblown sand particles on a high-speed train

C. Paz*, E. Suárez, C. Gil, M. Concheiro

School of Industrial Engineering, University of Vigo, Campus Universitario Lagoas-Marcosende, 36310, Spain

A R T I C L E I N F O

Article history:
Received 27 January 2015
Received in revised form
15 June 2015
Accepted 16 June 2015
Available online 2 July 2015

Keywords:
High-speed train
CFD
Aerodynamics
Windblown sand
Impact
Drag force

A B S T R A C T

This study develops an approach to evaluate the effect of particle impacts on the motion of high-speed trains in a sand-laden flow. For this purpose, external routines were implemented to extend the Discrete Phase Model (DPM) of the commercial simulation code ANSYS Fluent. Although the aerodynamics is an established concern in the design of high-speed trains, relatively few studies investigate the response of trains to demanding environments such as deserts. Several of the derived problems include potential effects on the aerodynamic performance or the wear of materials. Simulations with different values for the particle diameter, particle load and coefficient of restitution were performed. The analysis of the leading vehicle shows a greater impact probability on the train nose with small impact angles and high velocities on the sides, leading to a more pronounced wear of the surface. A logarithmic dependency of the drag coefficient with the particle diameter was also revealed, and a force reduction of 10% for each 0.2 decrease in the coefficient of restitution was noted. The results of the simulations confirm the feasibility of the presented methodology.

1. Introduction

The motion of high-speed trains and their response to aerodynamic effects derived from increased velocities, such as crosswinds (Hemida and Krajnovic, 2010; Schetz, 2001), slipstreams (Muld, 2012; Weise et al., 2006), pressure variations (Gawthorpe, 2000; Ko et al., 2012) or ballast flying (Jing et al., 2012; Kaltenbach et al. 2008), have long been an important field of research in the railway industry (Baker, 2014a, 2014b; Raghunathan et al., 2002). The majority of these studies are focused mainly on the evaluation of the fluid field around the train under different conditions through full-scale (Baker et al., 2013) or scaled experiments (Gilbert et al., 2013) and numerical simulations (Hemida et al., 2014), analysing more deeply certain areas, e.g., the wake (Bell et al., 2014) or the underbody (García et al., 2011). In recent years, the development in countries such as China or Saudi Arabia has led to the construction of new tracks crossing desert areas, in which windblown sand has become a matter of concern. The motion of high-speed trains in particle-laden flows is affected by many factors such as the risk of overturning caused by sandstorms (Qian et al., 2002), the premature wear of train elements (Woldman et al., 2012) or potential effects on the aerodynamic performance. In China, for example, many accidents have been reported. Other countries (e.g., Saudi Arabia) are dealing with this problem in new infrastructures, trying to avoid the plugging of the vents, the obstruction of moving elements or the damaging of the train surface. The collision forecast of sand particles on the train surface could establish a proper dimensioning and positioning of the ventilation and refrigeration grilles. Additionally, the impact angle points out areas with the highest risk of erosion, i.e., those areas requiring the use of specific countermeasures such as paintings or coatings.

These two mentioned problems (wear of materials and aerodynamic effects) are related to the exchange of energy during the impact between the particles and the train surface. This impact is feasibly modelled through computational methods. The physics of the collision is a complex and well-studied process (Hertz, 1896; Kosinski et al., 2014; Kuwabara and Kono, 1987). When two elastic bodies collide, their surfaces are deformed as a function of their momentum and mechanical properties. The elastic wave generated by the impact promotes the energy exchange between solids and the dissipation of energy into the surroundings in the form of sound or heat. Different expressions have been proposed to measure the characteristics of the impact such as the contact time or the exerted force (Antonyuk et al., 2010; Cowell et al., 2015).

From the point of view of computational fluid dynamics (CFD) simulations, two different strategies are generally accepted as the most suitable to evaluate this type of problem. On the one hand, the Eulerian–Eulerian (E–E) model treats both phases as

---

* Corresponding author. Tel.: +34 986 813 754.
E-mail addresses: cpaz@uvigo.es (C. Paz), suarez@uvigo.es (E. Suárez), chgil@uvigo.es (C. Gil), mconcheiro@uvigo.es (M. Concheiro).

http://dx.doi.org/10.1016/j.jweia.2015.06.008
0167-6105/

continuous flows interpenetrating each other. This approach has been followed by Xiong et al. (2011) in their study of the performance of high-speed trains under different levels of sandstorms. On the other hand, the Eulerian–Lagrangian (E–L) model solves the fluid flow using Navier–Stokes equations, and the solid particles are injected into the flow and are then tracked individually to calculate their trajectories inside the gas (Valentine and Decker, 1995). This strategy offers a more comprehensive picture of the particle flow interaction but requires more powerful computational resources (Zhang and Chen, 2007). For this reason, the size and distribution of the particles must be properly characterised and the injections must be reduced to the minimum necessary (Kaufmann et al., 2008).

In this paper, the E–L approach has been used to simulate the motion of a high-speed train in a sand-laden atmosphere. The objectives of this study are to quantify the force exerted on the train by sand grains (with different ranges of particle diameter, particle load and coefficient of restitution between particle and surface) and to determine the regions with a higher probability of suffering impacts, focusing on the leading vehicle, and the characteristics (angle, velocity and force) of these collisions.

Because of the complexity of the dynamics of the collision, the computational costs of the CFD calculation of the detailed process are prohibitive (Stevens and Hrenya, 2005). The strategy followed here involves a feasible macroscopic approach using the average force during the interval of time between two consecutive impacts, retaining the desirable accuracy.

The selected geometric model and the conditions of the simulation of the continuous phase are described in Section 2. In Section 3, the parameters for the particle tracking, including the characterisation of the sand grains, are detailed; additionally, the routines implemented for the assessment of the generated forces are provided. In Section 4, the contours of probability, normalised velocity, tangential velocity, angle and force of the impacts and the graphs for the drag force contribution are shown and discussed. Finally, the conclusions are summarised in Section 5.

2. Geometric model and boundary conditions

Although the main objective of this study is to evaluate the impacts on the leading vehicle, an entire train has been simulated to solve the continuous phase. The geometric model used is a simplified full-scale ETR 500, consisting of two power cars and six intermediate cars with a total length of 200 m. The geometry is a complete 3D model of the ETR500, including simplified bogies, in which all the details smaller than the surface cell size were not considered, and omitting singular elements such as the pantographs. This design, similar to the one used by Gil et al. (2008) and Muld et al. (2012), is considered sufficient to reproduce correctly the aerodynamics of a high speed train and to account for most of the relevant effects, such as the boundary layer development around the train or the turbulence due to the asymmetry of the bogies. For a more realistic representation, the model is mounted on a single-track ballast and rail (STBR) scenario (CEN European Standard, 2009). Since this study is focused on the analysis of the leading vehicle and the flow in this region would be almost unaffected by the presence of a second track, an STBR scenario was chosen to reduce the computational cost of the simulations. The computational domain is extended 8 H beyond the nose of the leading vehicle and 30 H from the train tail to the outlet. The height and width of the outer box which limits the domain are 10 H and 20 H, respectively (Hemida et al., 2014) (shown in Fig. 1).

Two different meshes were created to evaluate the influence of the grid resolution on the results. The scheme Cutcell, a utility within the software ANSYS Meshing, has been selected for both meshes with the intention of obtaining unstructured hexahedral grids aligned with the undisturbed upstream flow (Nemec et al., 2008). The coarse and fine meshes consist of 22 and 56 million elements with a cell size on the surface of the train of 0.012 H and 0.007 H, respectively. Based on the previous study of mesh convergence included in the paper of Paz et al. (2014), it is considered that both meshes have reached the convergence. However, both meshes were compared to evaluate their performance in the current geometrical model. A boundary layer around the model and the rails have also been included, leading to values of y^+ between 50 and 100.

Simulations were performed using the commercial software ANSYS Fluent. The k-epsilon Realisable viscous model (Cheli et al., 2010) was considered the most suitable for solving the continuous phase because of the balance between accuracy and computational cost. The inlet velocity was set uniformly to 300 km/h (83.33 m/s) and the outlet to atmospheric pressure. The STBR scenario and the ground were defined as moving walls with the identical speed as the air, reproducing the relative motion of the train and avoiding the use of sliding meshes (Flynn et al., 2014). To improve the convergence of the simulations, the cases were calculated first in steady-state and were afterwards changed to a transient mode. The time step was selected as a function of the oscillation frequency for a Strouhal number of 0.14 (Baker, 2010) and *H* as the characteristic length (see Fig. 1).

3. Particle impact approach

For the simulation of sand particles, the DPM model implemented in Fluent has been used. This E–L approach is considered appropriate for this purpose because the discrete volume fraction is much lower than 10% and the computational resources are sufficiently powerful (ANSYS Inc., 2012). To reduce the simulation costs, a one-way coupling between phases was selected, treating the sand grains as solid particles; no additional forces were

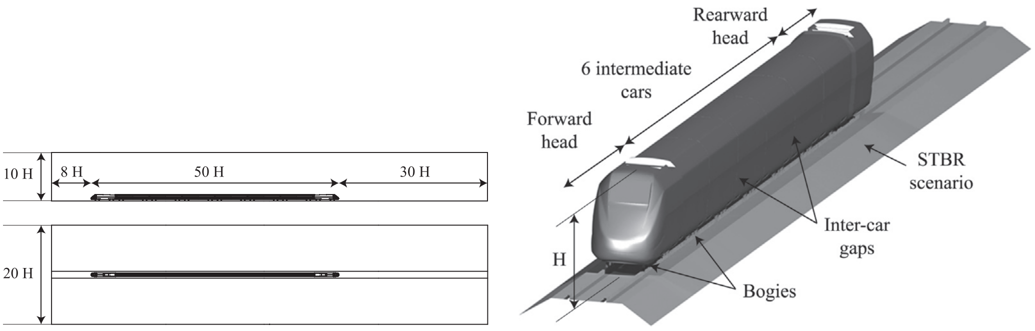


Fig. 1. Computational domain (left) and geometrical model (right).