Journal of Wind Engineering & Industrial Aerodynamics 167 (2017) 140–147

Contents lists available at ScienceDirect

Journal of Wind Engineering and Industrial Aerodynamics

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

# Numerical methodology for evaluating the effect of sleepers in the underbody flow of a high-speed train

C. Paz*, E. Suárez, C. Gil

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

## ARTICLE INFO

*Keywords:*
High-speed train
CFD
Aerodynamics
Underbody flow
Sleepers
Dynamic mesh
Layering
Sliding

## ABSTRACT

This paper presents an innovative numerical methodology based on dynamic meshes that allow for the 3D simulation of the motion of a high-speed train, reproducing the real movement and including the geometry of the sleepers in the calculation. The objective of this strategy is to provide a more accurate analysis of the underbody flow, which has become a matter of concern, especially because of the rising problem of the ballast flight. The proposed methodology, without sleepers, has been compared to the conventional procedure, consisting of a static train and a moving wall condition on the ground, with satisfactory results. Subsequently, the inclusion of the sleepers in the simulation revealed an increase in the drag coefficient of approximately 15%. Regarding the underbody flow, lower values of the minimum static pressure were observed in the entire profile from the ballast layer to the train and some strong peaks of the vertical component of the velocity were identified near the sleepers. Both factors could affect the process of ballast flight, which is estimated to be favoured by the presence of the sleepers. The results obtained demonstrated the applicability of the proposed methodology.

## 1. Introduction

In the past decades, the progress of the technology has led to an increase in the circulation speed of the trains and, consequently, the railway industry has to face new problems. Most of these problems such as the effect of the slipstream (Derkowski et al., 2014) or the ballast flight (Saussine et al., 2011) are strongly related to the development of the fluid field around the high-speed trains. Therefore, understanding of this fluid field is one of the main areas of research in this field (Baker, 2014a, 2014b; Sima et al., 2011).

On the one hand, the air flow either around the nose, in the development of the boundary layer along the sides and the roof or in the wake of the train has been widely investigated (Baker, 2010; Muld et al., 2012; Raghunathan et al., 2002). Due to the complexity and the elevated costs of the experimental campaigns (Weise et al., 2006), the wind tunnel scaled tests have become fundamental to the design and study of the high-speed trains. In this respect, the traditional tests (Willemsen, 1997), consisting of a static train placed on a flat ground or a track scenario, have lately been replaced by more sophisticated facilities that employ moving models on long straight tracks (Bell et al., 2015) or rotating rail rigs (Gil et al., 2010) that allow for the reproduction of the relative velocity between the train and the ground.

On the other hand, many investigations focus attention on the behaviour of the air in the underbody of the high-speed trains, which was until recently one of the aspects of the railway industry that was less investigated (Baker, 2010). However, the rising problem of the ballast flight has awakened the interest of the researchers in this region of the slipstream. For this purpose, experimental campaigns have been carried out in different countries by means of either full-scale measurements (Kaltenbach et al., 2008; Quinn et al., 2010) or wind-tunnel tests (Sima et al., 2008), using different types of transducers (e.g., pitot tubes, multi-hole probes, etc.) or even optical techniques such as particle image velocimetry (Jönsson et al., 2014). The objective is to characterize the flow field measuring the aerodynamic variables (pressure, velocity profile, loads on ballast) due to the passing of a train. Recently, Premoli et al. (2015) conducted wind-tunnel tests using a section of a real track, sleepers included, to evaluate the velocity profile near the ground and to identify the main parameters involved in the ballast flight phenomenon. The study concluded that a high velocity gradient starting from 2 to 3 cm above the sleepers characterises the flow over the track.

In both cases, the computational fluid dynamics (CFD) simulations have become an indispensable tool, validating the calculations with the experimental data to subsequently apply this CFD methodology to diverse scenarios (Cheli et al., 2008; Sima et al., 2008). Unfortunately, the limitations of computational resources make the employment of strategies to reduce the complexity of the geometry of the train and the track and the boundary conditions unavoidable, to reproduce the real

---

* Corresponding author.
*E-mail addresses:* cpaz@uvigo.es (C. Paz), Suarez@uvigo.es (E. Suárez), chgil@uvigo.es (C. Gil).

http://dx.doi.org/10.1016/j.jweia.2017.04.017
Received 14 October 2016; Received in revised form 20 March 2017; Accepted 22 April 2017

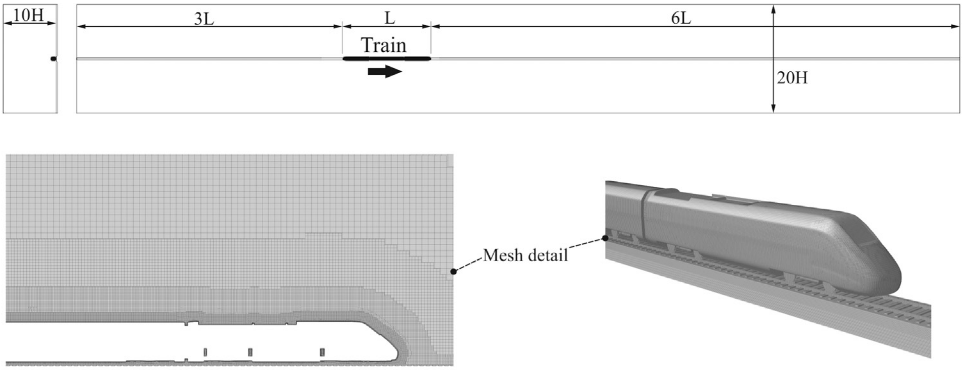

**Fig. 1.** Computational domain and details of the mesh.

conditions in the most accurate manner. In this respect, the most common strategy implies the use of a static train and a moving ground with the same velocity as the air flow. The wheel rotation is not commonly considered, and the track is represented by either a flat surface or a single track ballast and rail (STBR) (CEN, 2009]. Regarding these two ground configurations, the study by Bell et al. (2014) revealed significant differences in the flow structures between them. It states that the movement to the sides and in the streamwise direction of each vortex of the counter rotating pair generated in the wake is faster and with greater intensity in the case of flat ground, leading to less accurate results with respect to full-scale tests. The aforementioned simplifications are widely accepted in the simulation of the general aerodynamic performance or the train slipstream, being in good agreement with test data (Hemida et al., 2014; Paradot and Talotte, 2001). Furthermore, Zhang et al. (2016) have recently demonstrated that the rotation of the wheel does not play a significant role in the drag resistance of a high-speed train.

Nevertheless, reducing the scope of the study to the proximity of the ballast layer, the nature of the flow reproduced in the CFD might be distorted due to the simplifications. The use of a smooth flat surface without sleepers makes the section between the train and the track uniform, avoiding the narrowing and widening of the fluid path. Consequently, the longitudinal component of the velocity dominates the flow, leaving the lateral and vertical components underestimated. To cope with this inconvenience, some papers (García et al., 2011; Lazaro and Gonzalez, 2011) approximate the flow between the underbody and the track using a turbulent Couette flow with fully rough walls, trying to obtain an equivalent roughness of the track including the ballast and the sleepers. This strategy has provided an analytical approximation to the real situation but under the consideration of a 2D domain. Similarly, Rocchi et al. (2013) performed simulations setting different degrees of roughness for the lower wall in the pursuit of a value that fits the experimental data better. This method, however, depends strongly on the accuracy of the field measurements and requires a validation whenever the method is applied.

As an alternative, the current paper presents an innovative CFD methodology based on dynamic meshes that allow for the 3D simulation of the motion of a high-speed train, reproducing the real movement of the train and including the geometry of the sleepers in the calculation. Prior to this study, no evidence of any other validated numerical methodology including the railway sleepers in the simulation of a high speed train was found. The objective of this strategy is to provide a more accurate analysis of the underbody flow, determining the velocity profile near the track either in the presence of a sleeper or in the space between them, and estimating the effect on the stones lying in the ballast layer. All the results are compared with the results obtained from a simulation using the conventional methodology.

In Section 2, the proposed numerical methodology is described, including the geometrical model selected for the simulations, the dimensions of the computational domain and the characteristics of the mesh. The results of the study are subsequently shown in Section 3, divided into the following parts: validation of the method that is presented against the conventional one; analysis of the underbody flow in the presence of sleepers; and possible effects of the fluid flow on the ballast layer. Finally, the conclusions are summarized in Section 4.

## 2. Numerical modelling

### *2.1. Geometrical model and computational domain*

In this study, a full-scale 3D model of the ETR500 train mounted on a single-track ballast and rail (STBR) scenario with sleepers has been simulated. The STBR is a kind of scenario widely used in recent studies (Bell et al., 2016; Niu et al., 2017). It is demonstrated to improve the representation of the turbulence of the fluid field around a high-speed train thanks to a more realistic development of the pair of counter rotating vortices generated in the wake (Bell et al., 2014). The model followed a power head + intermediate car + power head configuration, and included inter-car gaps and realistic bogies with four wheels, even though the geometrical details smaller than the surface cell size were not considered, nor were the pantographs and other singular elements. The simplified short models of entire trains are commonly employed both in wind tunnel tests and numerical simulations (Gil et al., 2010; Krajnovic et al., 2008). The sleepers were designed with a section of 220×50 mm (height from the top of the sleeper to the ballast layer), and the separation between them was set to 600 mm (gap =380 mm).

The dimensions of the computational domain have been established as a function of the train length (L=67 m) and train height (H=4.125 m). The domain was extended 6 L beyond the nose of the power car and 3 L from the tail to the outlet. The width and height of the outer box are 20 H and 10 H respectively (See Fig. 1) (Hemida et al., 2014). These dimensions have been set considering the proposed methodology consisting of a dynamic mesh in which the train moves forward inside the static domain. Therefore, it must be guaranteed that the distance to the outlet at the beginning of the simulation is sufficient to capture the effects in the wake, and the distance to the inlet at the end of the movement allows for a correct development of the incoming flow.

### *2.2. Computational mesh*

A study on the influence of the mesh resolution has been carried out with the simulation of five different computational grids, ranging from 0.9 to 17 million cells. The simulations of the convergence analysis of this mesh were performed by the conventional method, considering the STBR as a moving wall with no sleepers. The scheme Cutcell, a hex-dominant mesh technique, was used to create unstructured hexahedral grids aligned with the undisturbed upstream flow (Nemec et al., 2008). This method uses a patch independent volume meshing approach and, due to the large fraction of hex cells, often produces better results than