Journal of Fluids and Structures 61 (2016) 249–261

Contents lists available at ScienceDirect

Journal of Fluids and Structures

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

Impact of ground and wheel boundary conditions on numerical simulation of the high-speed train aerodynamic performance

Jie Zhang [a,1], Jing-juan Li [a,1], Hong-qi Tian [a], Guang-jun Gao [a,*], John Sheridan [b]

[a] Key Laboratory of Traffic Safety on Track of Ministry of Education, School of Traffic & Transportation Engineering, Central South University, Changsha 410075, China
[b] Department of Mechanical and Aerospace Engineering, Monash University, Clayton, Victoria 3800, Australia

A R T I C L E I N F O

Article history:
Received 29 July 2014
Accepted 22 October 2015
Available online 29 December 2015

Keywords:
Moving ground condition
Rotating wheel
Aerodynamic performance
High-speed train

A B S T R A C T

In this paper, the aerodynamic performance of a high speed train with moving ground and rotating wheels (MG&RW) conditions has been investigated using Computational Fluid Dynamic (CFD). The numerical simulations under the condition of stationary ground and moving ground are also compared and discussed. To validate the accuracy of the mesh resolution and methodology, the CFD results are compared with the wind tunnel test results. The aerodynamic forces, unsteady and time average flow fields around the high-speed train are analyzed. The main aim of this study is to investigate how the moving ground and wheel conditions influence the numerical simulation results. The moving ground condition eliminates the effects of boundary layers of ground and rail track, which results in the velocity under the train being faster than in the stationary ground condition. As a result, the drag of every bogie and the pressure on the bottom surface calculated under moving ground conditions is found to be higher than that under the stationary ground condition. The wheel rotation boundary condition has little effect on the force distribution on the bottom surface of the train. However, at the bogie regions, it increases the velocity of airflow around and behind the wheel, causing a change of the pressure distribution and an increment of the wheels' drag. Consequently, the total drag of the train in moving ground with rotating wheel condition is nearly equal to that of condition with the moving ground considered. In addition, the total drag of bogies just takes up 10.4% in all in the stationary ground condition, and it accounts for 12.7% in the moving ground condition and 15.1% in the moving ground with rotating wheel condition.

© 2015 Elsevier Ltd. All rights reserved.

1. Introduction

With the rapid development of high-speed trains around the world, the investigation of three-dimensional flow around them has become of significant importance in the rail industry. The execution of proper aerodynamic design requires an extensive understanding of the relevant flow phenomena and aerodynamic performance. In recent years, there has been

* Corresponding author.
E-mail address: gjgao@csu.edu.cn (G.-j. Gao).

[1] These two authors contributed equally to this work.

http://dx.doi.org/10.1016/j.jfluidstructs.2015.10.006
0889-9746/© 2015 Elsevier Ltd. All rights reserved.

considerable work done that studies high-speed trains' aerodynamic performance. The basic tools used include full-scale tests, wind tunnel tests and computational fluid dynamics (CFD) (Xiao et al., 2013).

When the train runs, the movement of the train will include wheel rotation and the relative motion between the wind and the train, as well as the relative motion between the stationary ground and the moving train. Full-scale tests are undoubtedly the most reliable approach because they investigate on the real operation of the train. However, in the case of train aerodynamics, full-scale tests and wind tunnel tests are difficult (Baker, 2010). Full-scale measurements strongly depend on environment and very often a large number of runs have to be carried out to obtain the reliable results.

With regard to wind tunnel tests for the investigation of train aerodynamic performances, the velocity inlet condition is given to simulate the relative motion between the train and the wind. The wind tunnel tests are difficult because of the ground effect. In terms of the aerodynamic force measurement, the ground effect simulation will affect the results, and is difficult to be eliminated. Although some new technologies, such as installation of the moving ground and suction-from-below devices on the floor under the train have been found effective to reduce or eliminate the ground effect (Xiao et al., 2013; Baker, 1991, 2010; Kwon et al., 2001), the application of these technologies is limited due to their high cost and complex operation. Most wind tunnel tests have not considered the moving ground and rotating wheel conditions, as demonstrated in published papers (Orellano and Schober, 2006; Schober et al., 2010; Huang et al., 2013; Huang et al.,2012; Bell et al, 2014). In addition, the measurement and display of the flow field in wind tunnel tests require special techniques capable of qualitative and quantitative analysis.

CFD is widely used since it is capable of efficiently computing and visualizing the flow field around trains. Lots of research about the aerodynamic performance using CFD has been done previously (Miao and Gao, 2012; Krajnovic et al., 2012; Hemida and Krajnovic, 2010; Raghunathana et al., 2002; Diedrichs, 2008). Zheng and Yang, (2011) have investigated the aerodynamic performance of high-speed trains in open air using Detached-Eddy Simulations (DES). Yao et al. (2013) have researched the mechanism of vortex formation and evolution in the train flow field using DES. A large number of studies have been undertaken and reported in Zhang et al. (2011), Zhang and Xiong (2011) and He (2011). In these numerical simulations, the ground movement relative to the train is considered by giving the ground the same velocity as the inlet flow, but the wheel-rotation condition is usually ignored. According to some CFD results in numerical simulations of a high-speed train with bogies (Zhang and Xiong, 2011; Yao et al, 2012), we discovered that the total drag of the head car is bigger than the tail car, but the results are found to be contradictory to this in wind tunnel tests. In the research of this paper, we have previously found discrepancies between wind tunnel and CFD results of simulation under stationary ground condition that suggest the total drag of the head car could in fact be lower than the tail car.

Based on the discussion above, the details of aerodynamic performance have not been analyzed for the running train. And the difference of aerodynamic performance of trains tested under stationary ground condition and moving ground with wheels rotating condition has not been researched. The aim of this paper is to investigate the aerodynamic performance of trains in the moving ground and rotating-wheel conditions using CFD tools, and study how the moving ground and wheel conditions influence the numerical simulation results. The mechanism effect of the moving ground and wheel condition based on the numerical simulation results is analyzed. The numerical simulation of the wind tunnel test with stationary ground and with moving ground are investigated as case 1 and case 2. In case 3, the numerical simulation is performed under the moving ground with rotating wheel (MG&RW) condition to investigate the aerodynamic performance of the train and the impact of the wheel rotation condition on the results.

This paper is organized as follows. In Section 2, the geometry, the numerical method, meshes and boundary conditions are given together with the cases studied. The train aerodynamic forces in case 1 are then compared with the wind tunnel test results to validate the accuracy of the resolution of the mesh and methodology. In Section 3, the drag in each case is compared and analyzed. The mechanism of the moving ground and rotating wheel conditions and how they affect the numerical simulation results are then explained. Finally, conclusions are drawn in Section 4.

2. CFD analysis

2.1. Computational model and boundary conditions

The wind tunnel model and the numerical models are basically the same, being a simplified version of the CRH2 train, which contains bogies, wheels and windshields, as shown in Fig. 1. It is grouped as three cars (the head car, the middle car and the tail car). The model scale is 1/8th, so the total length of the train is $L=9.5516$ m and its projected cross section area is $A=0.175$ m^2. Moreover, to make accurate and independent force measurements for each car and to guarantee that the flow field of the train is not influenced, the internal and external windshields between adjacent cars were separated by nested form in wind tunnel tests (Huang et al., 2012), as illustrated in Fig. 1. To reduce any influence of the gaps on the computational results, the gaps between adjacent cars were retained in the computational model.

The CFD results are compared with the results from wind tunnel tests. The measurements were made in the second test section of 8 m $\times$ 6 m wind tunnel in China Aerodynamics Research and Development Center (CARDC). The test setup is shown in Fig. 2(a). To reduce the thickness of the approaching boundary layer, a fixed ground board with a rotating table device was installed especially for the high-speed train tests. The distance of the floor device to the lower wall of the wind tunnel is 1.06 m. Then the test section is transformed into the one for high-speed train tests, which is 4.94 m high and

Download English Version:

https://daneshyari.com/en/article/792294

Download Persian Version:

https://daneshyari.com/article/792294

Daneshyari.com