Journal of Wind Engineering & Industrial Aerodynamics 179 (2018) 102–117

Contents lists available at ScienceDirect

Journal of Wind Engineering & Industrial Aerodynamics

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

Assessment of the mesh refinement influence on the computed flow-fields about a model train in comparison with wind tunnel measurements

K.A. Weinman [a,*], M. Fragner [b], R. Deiterding [c], D. Heine [a], U. Fey [a], F. Braenstroem [d], B. Schultz [d], C. Wagner [a,e]

[a] German Aerospace Center (DLR), Institute of Aerodynamics and Flow Technology, Göttingen, Germany
[b] Hochschule Hannover, Fakultät I - Elektro- und Informationstechnik, Hannover, Germany
[c] University of Southampton, Engineering and the Environment, Southampton, UK
[d] Bombardier Transportation GmbH, Henningsdorf, Germany
[e] Technische Universität Ilmenau, Institute of Thermodynamics and Fluid Mechanics, Ilmenau, Germany

ARTICLE INFO

Keywords:
Aerodynamic drag
Mesh refinement
Mesh sensitivity
PIV

ABSTRACT

A consistent mesh refinement study, relating to the prediction of aerodynamic forces about an experimentally validated reference train geometry, is presented in this paper. The flow about a high-speed train has a multi-scale character which poses challenges for the design of computationally effective meshes. The purpose of this study is to assist in the development of guidelines for effective drag prediction of high-speed trains using numerical simulation. These guidelines should assist CFD practitioners by identifying the regions of the mesh that are critical for the correct estimation of drag as well as providing information on appropriate mesh characteristics, such as volume and surface element length scales. Numerical assessments are validated against an experimental drag measurement program and the extent to which RANS is sufficiently predictive for industrial design is discussed. The results obtained in the work suggest that the mesh about the train nose is essential for the proper assessment of the aerodynamic drag acting on the vehicle.

1. Introduction

Stringent safety requirements over a wide range of operational conditions are applied to modern high speed trains. An understanding of the aerodynamic forces acting on a vehicle is mandatory, especially under crosswind conditions, in order to construct useful operational safety constraints. The measurement of force coefficients for full-scale vehicles is optimal but expensive, and normal practices are geared towards the use of small-scale models that can be tested inexpensively in wind tunnel experiments or by using full-scale-in-service vehicles (Baker, 2010). Experimental methods are limited in scope with respect to the study of questions such as the optimization of vehicle shape over a range of design parameters. In comparison, computational methods have the potential to provide detailed flow information at a cost that is comparatively inexpensive over a much wider range of operational conditions. For example these methods can be used to determine optimal shape forms in terms of stability and drag constraints.

The use of computational methods to assess the aerodynamic loading on trains has been recognized by the transport industry. For example the German standard EN 14067–6 (DB Netz AG, 2010) permits evaluation of aerodynamic forces by means of computational fluid dynamics (CFD) simulations for full-scale or reduced model geometries. The guidelines for CFD in EN 14067–6 using RANS (Reynolds-averaged Navier-Stokes) equations are stringent and give a specific error criterion that CFD calculations must satisfy. In particular, the standard requires that computed integral forces cannot be accepted for certification work if variations against accepted reference values (i.e. experiment) differ by more than three percent. A major challenge in satisfying EN 14067–6 requirements is due to the multi-scale nature of the flow problem which is characterized by a large range of energetically significant flow scales. Small-scale geometrical features of a train, for example the underflow region between the track and the train base (Sima et al., 2008), inter-car gaps and bogie cavities, can generate unsteady flow structures which interact with larger flow scales and thereby influence the development of the aerodynamic forces acting on the train. The underflow region contains numerous complex flow phenomena and is characterized

* Corresponding author.
E-mail addresses: keith.weinman@dlr.de, keith_weinman@hotmail.com (K.A. Weinman).

https://doi.org/10.1016/j.jweia.2018.05.005
Received 11 December 2017; Received in revised form 4 May 2018; Accepted 6 May 2018

typically by regions of flow separation driven by both geometry and incipient pressure gradient effects, together with cross-sectional area changes due to the underside geometry (e.g. inter-car gaps). In addition, the wake region (Muld, 2012) is dominated by vortex shedding events which contribute to the complexity of the modeling problem. Another contribution to the train aerodynamic force balance is provided by a steady vortex system originating from the front nose of the train (Baker, 2010, 2014; Hemida and Krajnovic, 2010). The front nose also contributes significantly to the train's operational drag penalty. Fig. 1 presents a breakdown of contributions to the total drag budget for a full-scale train under normal operating conditions in the absence of a cross-wind.

Some caution must be taken in comparing this figure with scaled-model wind-tunnel data since important differences may exist, for example Reynolds number equivalence is often not possible. The figure serves, however, as a useful indicator of where the critical contributions to the operation drag budget are found. Pressure drag over the train and tail dominate. The next largest component is the total skin friction drag over the train, which can be expected to scale linearly with train length. Fig. 1 implies that improvement in the prediction of head and tail drag as well as the skin friction drag will assist in the accurate assessment of operating costs for trains.

Computational assessments of the flow about a train have been traditionally undertaken on the basis of well-established RANS methods. The results of these examinations have not been totally satisfactory. Weinman et al. (2013) and Fragner et al. (2015) compared computational estimates of integral forces and moments using well-resolved meshes against the NGT2 experiment of Haff et al. (2012). Computations were performed for Reynolds numbers over the range $\mathscr{R} \in [250000, 750000]$ with cross-wind conditions of up to 30°. The computed integral force and moment coefficients, particularly drag, demonstrated differences against experimental measurements of up to 15 percent. Unsteady turbulent flow resolving methods demonstrate improved predictive capabilities over a wide range of unsteady flows when compared with RANS methods. Hemida and Krajnovic (2010) used Large Eddy Simulation to examine the flow over a high-speed train under cross-wind conditions. Morden et al. (2015) also investigated surface pressure loads using Delayed Detached Eddy Simulation (DDES) and a wind tunnel model. As with LES, improvements are often marginal and the efficiency of RANS methods – provided an appropriate turbulence model is available – can make it difficult to justify these computationally demanding approaches over RANS. Morden et al. obtained good results in computing vehicle surface pressure loads with RANS when using the Menter-SST model (Menter et al., 2003). Fragner et al. (2015) and Fragner and Deiterding (2016) validated highly resolved RANS, URANS, LES, Delayed Detached Eddy Simulation (DDES), and Lattice Boltzmann (LBM) methods against the NGT2 experiments. Their work demonstrated

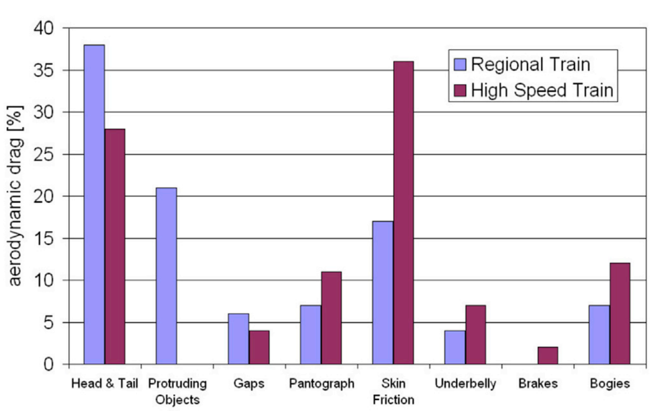

Fig. 1. Contributions to the total drag of typical trains (Orellano, 2010).

improvement for DDES, LES and the LBM methods against conventional RANS in the prediction of the aerodynamic pitching moment, however differences against the measured drag were still unsatisfactory. The LBM method used returned closer agreement with experiments and demonstrated a speedup of ~ 16 against the competing finite volume methods, due to a novel adaptive meshing technique (Deiterding, 2011) and the explicit calculation of the LBM partial density distribution advection step. However LBM methods remain restricted at present to the low Mach number range. Unsteady methods such as LES, DDES and LBM can present challenges for use in the industrial environment. Significant computational resources are usually required. These methods have not yet demonstrated a level of improvement over RANS methods in the prediction of vehicle aerodynamic loads that would justify their use. The earlier observation of Sima et al. (2008), who noted that traditional RANS methods would retain their importance for the foreseeable future, still remains valid today.

Investigation of the behavior of RANS methods when applied to the analysis of flows around trains is relevant for current industrial applications. A critical component of a RANS calculation is the design of the computational mesh. Mesh requirements for LES are often stated for simple flows and Spalart (2001) has provided a detailed guide for the generation of appropriate grids for hybrid RANS-LES methods such as DES. Detailed recommendations applicable in the design of RANS meshes for flows about trains appear to be absent in the literature. In this paper the influence of mesh resolution on the computed drag force of a model train is examined against experimental validation data and initial recommendations for the design of the mesh are provided. As part of this present study a series of experiments under low-speed flow conditions for a scaled model configuration were conducted using the cross-wind facility of the Simulation Center of Aerodynamic Research in Transportation (SCART) at the DLR Institute of Aerodynamics and Flow Technology located in Göttingen. This facility has been used successfully for the experimental measurements of aerodynamic forces acting on a range of ground based vehicles (Haff et al., 2012). The paper is organized as follows. Selection of the train model and experimental layout is provided in Section 2. Wind tunnel experiments are described in Section 3. An overview of the CFD geometry is given in Section 4 and Section 5 provides a description of the numerical tools used. Discussion on differences between the computed and experimental force coefficients, surface pressure distributions and wake flow structure are found in Section 6. Computational efficiency and conclusions are discussed in Sections 7 and 8, respectively.

2. Train model and experimental setup

For this paper a model consisting only of the main train aerodynamic surfaces and the wind tunnel is considered. Additional geometrical features, such as inter-car gaps and bogie cavities, are not considered in the present investigation: the goal is to examine the influence of mesh resolution using the simplest representative geometry. Fig. 2 illustrates the wind tunnel model as well as the coordinate system used for both the experiment and numerical calculation.

The reference system is aligned with the stationary inertial reference frame of the wind tunnel. Further details of the model can be seen in Fig. 2. The model segments are fastened to a hollow steel $0.1 \times 0.1\ \mathrm{m}^2$ beam. Two Kistler Piezo-electric sensors of type 9317-B are mounted between two 0.02 m thick steel plates, and two of these assemblies are located with a separation distance of 1.2 m on the steel beam. The lower parts of the assemblies are fixed to a $1.6\ \mathrm{m} \times 0.2\ \mathrm{m}$ steel plate of 0.02 m thickness, which is embedded into the wooden wind tunnel splitter plate (3.302 m in length), using posts (see Fig. 2). The plate extends from the left tunnel wall to the right tunnel wall. The mono-block model consists of two end cars connected via the steel beam at 1/15 scale of a full-sized vehicle. The model scale was chosen on the basis of a numerical study on the wind tunnel blocking effect as