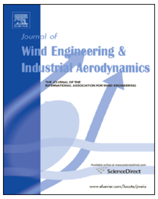
J. Wind Eng. Ind. Aerodyn. 136 (2015) 138–150

Contents lists available at ScienceDirect

Journal of Wind Engineering
and Industrial Aerodynamics

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



Bridge deck flutter derivatives: Efficient numerical evaluation exploiting their interdependence

F. Nieto [a,*], J.S. Owen [b], D.M. Hargreaves [b], S. Hernández [a]

[a] School of Civil Engineering, University of A Coruña, Spain
[b] Faculty of Engineering, University of Nottingham, UK

ARTICLE INFO

Article history:
Received 4 April 2014
Received in revised form
3 November 2014
Accepted 5 November 2014
Available online 25 November 2014

Keywords:
Computational fluid dynamics
Bluff body aerodynamics
Flutter derivatives
Rectangular cylinder
Streamlined deck sections

ABSTRACT

Increasing the efficiency in the process to numerically compute the flutter derivatives of bridge deck sections is desirable to advance the application of CFD based aerodynamic design in industrial projects. In this article, a 2D unsteady Reynolds-averaged Navier-Stokes (URANS) approach adopting Menter's SST k-ω turbulence model is employed for computing the flutter derivatives and the static aerodynamic characteristics of two well known examples: a rectangular cylinder showing a completely reattached flow and the generic G1 section representative of streamlined deck sections. The analytical relationships between flutter derivatives reported in the literature are applied with the purpose of halving the number of required numerical simulations for computing the flutter derivatives. The solver of choice has been the open source code OpenFOAM. It has been found that the proposed methodology offers results which agree well with the experimental data and the accuracy of the estimated flutter derivatives is similar to the results reported in the literature where the complete set of numerical simulations has been performed for both heave and pitch degrees of freedom.

© 2014 Elsevier Ltd. All rights reserved.

1. Introduction

Long span bridges are prone to aeroelastic phenomena such as vortex induced vibrations, flutter, or buffeting. In fact, safety against flutter instability is one of the fundamental requirements in long span bridge design. If the wind speed exceeds the critical flutter speed of the structure, self-excited oscillations of the deck would rapidly amplify causing the collapse of the bridge.

The most widely used method for the identification of the flutter critical wind speed is Scanlan's approach, developed in the 1970s (Scanlan and Tomko, 1971), where a set of semi-empirical functions, named flutter derivatives, must be identified in order to define the motion-induced aerodynamic load acting on the bridge deck (Bartoli and Mannini, 2008). Traditionally, the identification of flutter derivatives has been conducted by means of wind tunnel tests of sectional models of bridge decks. The application in recent years of numerical methods in the identification of flutter derivatives aims at avoiding expensive and cumbersome experimental campaigns which are the standard approach in industrial applications currently.

In Computational Fluid Dynamics (CFD) modeling the flutter derivatives identification can be done following two different approaches (Fransos and Bruno, 2006). The first one requires the simulation of the forced harmonic oscillations in pitch and heave degrees of freedom. Then, the flutter derivatives are identified from the amplitude and phase relationships between the imposed displacement and the induced aeroelastic forces. The second method, based on indicial theory, requires simulating an abrupt displacement of the body immersed in the flow, which causes non-stationary forces. The flutter derivatives can then be computed from the ratio between the Fourier transforms of the step-response non-stationary forces and the prescribed step-input displacement. The methodology, based on the simulation of forced oscillations, has been, by far, more widely used than the one based on the indicial approach despite the apparent efficiency of the indicial function approach.

Focusing on applications of the harmonic forced oscillations approach, the trend in the 1990s and early 2000s has been developing in-house CFD solvers based on the finite-difference, finite element, finite volume, or discrete vortex methods. The references in the literature are numerous and some examples, without intending to be exhaustive are: Mendes and Branco (1998), Larsen and Walther (1998), Morgenthal and McRobie (2002), Xiang and Ge (2002), Vairo (2003), Jeong and Kown (2003), Frandsen (2004), Zhu et al. (2007) and Zhu et al. (2009). Developing in-house software has obviously been a barrier for the application of numerical methods in industrial bridge design problems due to its scientific complexity and the required labor and financial resources. Therefore more recently the

---

* Corresponding author: Tel.: +34 981 167000; fax: +34 981 167170.
E-mail address: felix.nieto@udc.es (F. Nieto).

http://dx.doi.org/10.1016/j.jweia.2014.11.006
0167-6105/© 2014 Elsevier Ltd. All rights reserved.

focus has been put on applying general purpose commercial finite volume solvers in bridge aerodynamics problems. An early application was authored by Bruno et al. (2001) who used FLUENT for studying the aerodynamic response of a static box deck and the effect of section details such as fairings and barriers. Fluid-structure interaction problems have been addressed more recently. In Ge and Xiang (2008) both in-house solvers and the commercial code FLUENT are applied, depending on the chosen approach for turbulence modeling. Sarwar et al. (2008) obtained the flutter derivatives of a bridge deck section and high aspect ratio rectangular cylinders by means of 3D Large Eddy Simulation (LES) using FLUENT. Huang et al. (2009) also used FLUENT to compute the flutter derivatives of the Great Belt Bridge and the Sutong Yangtze cable-stayed bridge. Starossek et al. (2009) employed the commercial software COMET to obtain the flutter derivatives of 31 different bridge sections, including experimental validation for a subset of nine sections tested in a water tunnel. Bai et al. (2010) used a combination of in-house code and ANSYS-CFX commercial software for computing force coefficients and flutter derivatives of various 3D deck sections. Huang and Liao (2011) used FLUENT to simulate forced oscillations of a flat plate and a bridge deck containing a linear combination of a set of frequencies. Also, Brusiani et al. (2013) employed FLUENT to compute the flutter derivatives of the Great Belt Bridge using a different turbulence model than Huang and co-workers. Of particular interest is the growing use of open source general CFD solvers. In Sarkic et al. (2012), the open source code OpenFOAM is applied to numerically replicate the wind tunnel test for identifying the force coefficients and flutter derivatives of a box deck cross-section. A more recent application by some of the authors of the former reference can be found in Sarkic and Hoffer (2013) where the LES turbulence model is applied to the same box deck.

CFD applications based on indicial functions are scarce in spite of its potential. In Bruno and Fransos (2008) it has been remarked that in this method just a single simulation for each degree of freedom is required to identify the complete set of flutter derivatives and that only the transient flow needs to be simulated. Thus, this approach is less demanding in computational resources than the classical forced oscillation-based method. On the other hand, the problem is particularly challenging from the CFD simulation perspective. Early applications are Lesieutre et al. (1994) who simulated the motion of a wing in the frame of an application to aircraft manoeuvers and Brar et al. (1996) who applied the Finite Element Method to obtain the flutter derivatives of an airfoil and a rectangular cylinder. A modified smoothed indicial approach was further developed in Fransos and Bruno (2006) and Bruno and Fransos (2008) who used FLUENT to obtain the flutter derivatives of a flat plate of finite thickness and studied also the effect of the Reynolds number on the flutter derivatives. The indicial approach has also been applied in the frame of a probabilistic study of the aerodynamic and aeroelastic responses of a flat plate (Bruno et al., 2009). More recently, Zhu and Gu (2014) have presented a method to extract the flutter derivatives of streamlined bridge decks, even if the application of the modified indicial approach to bluff bodies remains questionable.

From the previous review of the state of the art regarding applications of CFD in the design of long span bridges, the main reasons why numerical simulations are not being generally applied in bridge design in the industry to complement wind tunnel tests need to be discussed. Developing and upgrading in-house software is a complex task and requires highly skilled personnel and substantial funding. Consequently, it can only be achieved by a small number of organizations in the world. The increasing use of commercial software in recent years is making it easier to access the required technology. However, the cost of licenses, particularly for running massively parallel simulations, in many cases prevents the extensive use of CFD in design problems. This circumstance has made particularly appealing the use of open source solvers for both industry and academia, and open source software has already been applied in bridge design problems. Besides this, the increasing number of published successful simulations in bridge related problems means that CFD techniques are nowadays more mature and therefore more robust and reliable.

In spite of the dramatic improvements in computational power and access to cluster technology of recent years, the computer power demands linked with modeling complex fluid-structure interaction problems remains a key issue. In this respect, any method or technique which allows decreasing computational demands would facilitate incorporating CFD-based design in bridge engineering design. A number of researchers have proposed explicit relationships between flutter derivatives which have proved to be reliable for streamlined bridge decks such as Matsumoto (1996), Scanlan et al. (1997), Chen and Kareem (2002) or Tubino (2005). The application of these formulae allows the number of computer simulations for obtaining the flutter derivatives to be reduced to just half of the number required following the standard approach based on forced harmonic vibrations in heave and pitch degrees of freedom. To the authors' knowledge the aforementioned approach has not been applied in CFD-based studies to date.

The aim of the current piece of research is to propose a cost effective, and therefore efficient, computer-based approach for obtaining force coefficients and flutter derivatives of bridge deck box sections which could be used in industrial applications where the shape of different bridge deck designs could be numerically optimized. Consequently, a 2D URANS strategy is proposed, using the general purpose open source CFD solver OpenFOAM v2.1.1 in combination with the explicit relationships between flutter derivatives mentioned above. The more demanding 3D Detached Eddy Simulation (DES) or LES approaches, in spite of their superior accuracy, have not been considered in this work since they would pose additional challenges in terms of higher computer power demands and model setup.

A rectangular cylinder showing a separated and reattached time-averaged flow pattern has been selected as one of the case studies for the computation of the flutter derivatives. In particular, a ratio $B/H=4.9$ rectangular cylinder (B is the prism width and H is the height) was chosen in order to replicate an existing sectional model at the wind tunnel of the University of Nottingham. In the literature, the number of published references, both experimental and computational, dealing with the response of $B/H=5$ rectangular cylinders is plentiful, to a great extent thanks to the BARC initiative (Bruno et al. 2014). Taking into account the expected minimal differences between the aerodynamic response of $B/H=4.9$ and $B/H=5$ rectangular cylinders, for the sake of the efficiency of means in research, the authors have considered that the existing literature on 5:1 rectangular cylinders is adequate for the validation of the force coefficients and the flutter derivatives of the $B/H=4.9$ rectangular cylinder at 0° angle of attack. However, in the case that additional numerical studies would require validation against experimental data outside the range found in the literature, further wind tunnel tests could readily be conducted using the existing $B/H=4.9$ sectional model.

The second application case has been the G1 generic box section described in Scanlan and Tomko (1971) and Larsen and Walther (1998). The modern practice in long span bridge design has incorporated box deck cross-sections as the most common choice for these challenging structures. There are several reasons for this: a good aerodynamic and aeroelastic response characteristic of streamlined cross-sections, high torsional stiffness, construction economy and, in many cases, superior aesthetic value compared to truss girders. Recent examples of applications comprising box decks are the Forth Replacement Crossing in the United Kingdom, the Normandy Bridge and Millau Viaduct, in France, the Sutong Bridge in China or the Russky Bridge in Russia, amongst many others.

In the first part of this article, the fundamental formulation and the numerical approach adopted, along with the computational

Download English Version:

https://daneshyari.com/en/article/292470

Download Persian Version:

https://daneshyari.com/article/292470

Daneshyari.com