Journal of Wind Engineering & Industrial Aerodynamics 169 (2017) 30–38

Contents lists available at ScienceDirect

Journal of Wind Engineering & Industrial Aerodynamics

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

Numerical explorations of the limit cycle flutter characteristics of a bridge deck

Xuyong Ying, Fuyou Xu *, Mingjie Zhang, Zhe Zhang

School of Civil Engineering, Dalian Univ. of Technology, Dalian 116024, China

ARTICLE INFO

Keywords:
Bridge
Limit cycle flutter
Fluid-structure interaction
Numerical simulation
CFD

ABSTRACT

The aeroelastic responses of a bridge deck was numerically simulated using a fluid-structure interaction (FSI) model, whose accuracy was verified by the flutter responses of a thin plate with theoretical solutions. With the increase of attack angle, the deck section becomes much blunter, which leads to be more prone to limit cycle flutter (LCF). The LCF phenomena for a bridge deck were simulated by the present numerical model. The satisfactory accuracies of the numerical simulations are verified by comparing with the experimental results. The numerically calculated results show that the steady-state responses of the vertical-torsional coupled LCF are independent of the initial excitation conditions. The structural damping has remarkable influence on the critical wind speed and the LCF amplitude. The phase angle difference between the torsional and vertical motions slightly increases with the increase of wind speed. The developed numerical simulation approach can help to serve as a building block for developing an overall analysis framework for investigating the LCF characteristics of long-span bridges.

1. Introduction

For long-span flexible bridges, flutter is the most dangerous wind-induced divergent vibration, which should be strictly prohibited. Conventionally, in order to determine the critical flutter state of bridges, a large amount of flutter analyses, e.g. the Great Belt Bridge (Larsen and Walther, 1997), the Akashi-Kaikyo bridge (Katsuchi et al., 1998), the Golden Gate bridge (Jain et al., 1998), and the Messina Strait Bridge (Diana et al., 2004), have been conducted based on Scanlan's linear self-excited forces model (Scanlan and Tomko, 1971), in which the bridge deck undergoes harmonic vibration with net zero damping at the critical flutter state, and further increase of wind speed will result in catastrophic divergent vibration. This kind of flutter is classified as the divergent type.

According to the Scanlan's linear model, the self-excited forces are the linear functions of deck motions and flutter derivatives. Actually, the self-excited forces inevitably contain nonlinear components due to the bluff configurations and large amplitudes, and therefore, nonlinear aeroelastic phenomenon may be observed for some cases of bridge flutter. For numerous wind tunnel tests on bluff and/or streamlined deck sections with large attack angles, when the wind speed is sufficiently high, the decks experience limit cycle oscillations (LCOs), rather than the divergent flutter (Xu and Chen, 2009; Long, 2010; Zhu and Gao, 2015). This kind of oscillation can be classified as the limit cycle flutter (LCF), and the steady-state vibration amplitudes vary with wind speeds. Similar aeroelastic phenomenon can be observed in the field of aeronautics and astronautics engineering, e.g., in a wide variety of aircraft during flights (Cunningham, 2003), in wind tunnel tests (Majid and Basri, 2008; Amandolese et al., 2013), and in numerical simulations (Tang et al., 2003; Wang and Zha, 2011). For aircraft wings, large attack angles can induce the flow separation from the suction surface, and they are prone to experience LCOs due to the aerodynamic nonlinearities.

Recently, the LCF phenomena for bridge decks have received increasing attentions. Chen and Kareem (2004) examined the efficacy of the tuned mass damper (TMD) in controlling the self-excited vibration, and the results showed that the structural damping has negligible effect on the divergent flutter, while it is relatively effective for increasing the critical wind speed of LCF. Zhang (2007) presented a single-degree-of-freedom (SDOF) nonlinear aerodynamic model, by which the phenomenon of LCF was explained. However, only the torsional mode was considered. Xu and Chen (2009) investigated the LCF characteristics of a Π-type blunt deck section via wind tunnel tests, and found that the flutter forms (divergent flutter/LCF) were related with the section sharps, the relative ratios of mode frequencies, masses and moment of inertia. Zhu and Gao (2015) carried out a series of wind tunnel tests on several typical bridge deck section models, and the characteristics of the vertical-torsional coupled LCF were investigated.

* Corresponding author.
E-mail address: fuyouxu@hotmail.com (F. Xu).

http://dx.doi.org/10.1016/j.jweia.2017.06.020
Received 26 August 2016; Received in revised form 27 June 2017; Accepted 27 June 2017

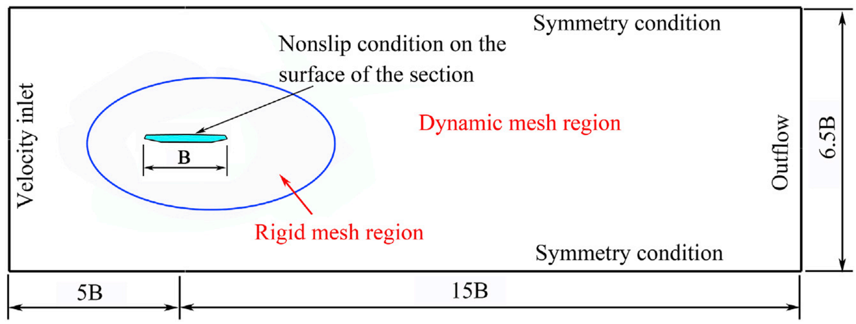

Fig. 1. Computational domain and boundary conditions.

As a consequence of the rapid advancement of computation power and improvement of computational fluid dynamics (CFD) technology, the numerical simulation approach provides an effective way to study the aeroelastic behaviors of flexible bridges. Fujiwara et al. (1993) calculated the flow field around elastically-supported edge beam cross-sections by using the finite-difference method (FDM); the vortex-induced vibration (VIV) phenomenon were successfully simulated and the onset wind speed predictions agreed well with the experimental results. Robertson et al. (2003) modified a two-dimensional (2D) spectral fluid solver in order to incorporate a body undergoing vertical and torsional motion, by which the fluid-structure interactions of a bridge deck were investigated. The effectiveness of the numerical approach was validated by comparing with the experimental results. Frandsen (2004) numerically investigated the wind-induced motion of 2D bridge decks by using finite element method (FEM), in which the bridge deck was idealized as lumped mass, spring and dashpot system. They concluded that the flutter instabilities for sharp edge bridge decks were insensitive to the turbulence and the modeling of a 3D flow structure. Although the VIV and divergent flutter of bridge decks have been well investigated by using numerical approaches, their applications to the LCF of bridge decks have not been documented.

The nonlinear LCF phenomena cannot be reasonably explained by the traditional linear analytical framework that used for the divergent flutter. In order to deepen the study of LCF phenomena of bridge decks, some research works may be required. For a specific design scheme, we need estimate the occurrence probability of a LCF by using experimental tests and/or numerical simulations. If a LCF occurs, it is also significant to study the characteristics of the varying amplitude with the wind velocity, and to determine how the oscillation is affected by modal parameters and wind field conditions. Therefore, the extensive investigations on the LCF of models are necessary to be conducted in advance to predict or estimate the aeroelastic response of real bridges, and further to address the flutter instability behavior of long-span bridges.

In the present study, the LCF characteristics of a bridge deck are comprehensively investigated by a fluid-structure interaction (FSI) numerical model. The simulation accuracy of the FSI model is verified by using a thin plate section by comparing the simulated results with the theoretical solutions. At various initial attack angles with different wind speeds, the dynamic responses of 2D deck model are numerically calculated. The vibration amplitude, frequency, and phase angle difference between torsional and vertical motions are analyzed, and compared with the experimental results. Under different initial excitation conditions and structural damping ratios, the LCF responses are also calculated and analyzed. Some significant conclusions are drawn and summarized.

2. Descriptions of the numerical model

2.1. Governing equations for fluid

The incompressible, unsteady, 2D air flow with moving boundaries can be modeled by means of the Reynolds Averaged Navier-Stokes (RANS) equations. For the numerical aeroelastic simulation that contains dynamic meshes, the accurate simulation for the interactions between air flow and moving deck section is an important requirement. In the present study, the governing equations are given in Arbitrary Lagrange-Euler (ALE) formulations, which accommodate the moving boundaries and any subsequent deformation of the underlying discrete meshes. By introducing the grid velocity u_{mj} of the moving mesh, the ALE formulations for the mass and momentum of incompressible fluid can be expressed as

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho (u_j - u_{mj})}{\partial x_j} = 0 \tag{1a}$$

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial \rho (u_j - u_{mj}) u_i}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\mu_{eff}\frac{\partial u_i}{\partial x_j}\right) + S_i \tag{1b}$$

where u_i or u_j, ρ, x_j and p are the fluid velocity components, the fluid density, the Cartesian spatial coordinates and the fluid pressure, respectively; S_i denotes the additional momentum source contributions, if any; μ_{eff} is the effective viscosity which includes laminar and turbulent contributions (Hassan et al., 2010). In the RANS approach, the turbulence viscosity is modeled by the SST $k-\omega$ model (Menter, 1994). Ying et al. (2012) conducted comprehensive simulations of the unsteady flow around rectangular cylinders by using six typical RANS turbulence models, i.e. standard $k-\varepsilon$model, RNG $k-\varepsilon$ model, realizable $k-\varepsilon$ model, standard $k-w$ model, SST $k-w$ model, and RSM model. Consequently, the SST $k-\omega$ model was found to be the best choice among various RANS models, and it is accurate enough to be suitable for many practical problems. Further details on the implementation of the SST $k-\omega$ model can be referred to Menter (1994).

The *ANSYS FLUENT* adopted in this study uses the finite-volume method (FVM) to solve the fluid governing equations. The ALE formulations of governing equations enable the conservative fluid calculations with mesh adaptation in time. The discretization method for the RANS governing equations remains unchanged from the general application of FVM (Schneider and Raw, 1987). The second-order implicit scheme and upwind scheme are used for the time and spatial discretization, respectively. The SIMPLE (semi-implicit pressure linked equations) algorithm is used for solving the discretized governing equations.

2.2. Computational domain and mesh arrangement

The computational domain and boundary conditions are schematically shown in Fig. 1 for a 2D *x-y* slice. Due to large vibration amplitudes, the wide rectangular computational domain is set as $15B \times 5B$ m. The boundaries are sufficiently far away from the sections so as to eliminate the flow obstacle effect on the inflow and outflow boundary conditions. At the inflow boundary, the flow with a low turbulence intensity of 0.5% is used. The non-slip condition is used for the section surfaces, and the symmetry condition is used for the top and bottom surface of the domain. It can be approximately considered that the flow is fully developed at the outlet boundary. Considering the mesh number should be as low as