Applied Ocean Research 75 (2018) 100–116

Contents lists available at ScienceDirect

Applied Ocean Research

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

Numerical investigation of hydrodynamic load on bridge deck under joint action of solitary wave and current

K. Qu[a,b], H.S. Tang[b,*], A. Agrawal[b], Y. Cai[b], C.B. Jiang[a]

[a] School of Hydraulic Engineering, Changsha University of Science & Technology, Changsha 410114, China
[b] Department of Civil Engineering, City College, City University of New York, New York 10031, USA

ARTICLE INFO

Article history:
Received 17 May 2017
Received in revised form 17 February 2018
Accepted 23 February 2018
Available online 4 April 2018

Keywords:
VOF
SIFUM
Bridge deck
Solitary wave
Current
Air vent

ABSTRACT

During the past few decades, there have been many instances of significant damage to coastal infrastructure, especially bridges, due to ocean waves generated by hurricanes and tsunamis. Since ocean waves and currents co-exist and twist with each other in natural marine environments and their interaction may result in more severe damaging waves, taking both of them and their interaction into account is important in better understanding of damage processes of coastal bridges. This paper conducts a numerical investigation on hydrodynamic load on a bridge deck due to joint action of solitary waves and currents. Effects of prominent factors including current velocity, submersion depth, wave height, and water depth have been studied. Efficiency of air vents in reducing the hydrodynamic load has also been discussed. The numerical investigation indicates that, in a linearly pattern, a current in the wave direction leads to a higher maximum of the hydrodynamic force in the horizontal direction, and a current in the opposite direction results in a lower maximum. However, the behaviors of other characteristics of the force, including the maximum of its vertical component and the minimums of its horizontal and vertical components, become complicated and highly nonlinear because of water overtopping on the deck. In addition, a current can play a pronounced role, either positive or negative, in efficiency of air vents in reducing the hydrodynamic load. It is anticipated that the findings in this paper will enhance our understanding on mechanism of bridge damage by waves and may also be useful in design of future coastal bridges.

1. Introduction

Coastal infrastructure is vulnerable to extremely large surges and waves generated by hurricanes and tsunamis accompanied by elevated water levels and high speed currents. During the past few decades, such surges and waves have caused significant damage to inshore and offshore infrastructure, especially coastal bridges around the world. As reported in [1], in 2005, Hurricane Katrina caused serious damage to the transportation infrastructure in the Gulf Coast region, resulting in the collapse of 44 bridges. The overall cost to repair or replace these damaged bridges has been estimated to be over one billion US dollars. In 2008, Hurricane Ike severely destructed 53 bridges in the Houston/Galveston region of Texas [2]. In 2012, Hurricane Sandy 2012 led to damage in 50 billion US dollars to coastal infrastructure, making it the second-costliest cyclone to hit the US since 1900 [3]. The 2011 Japan tsunami resulted in heavy damage up to seven miles inland and destroyed more than 162,000 buildings and more than 300 bridges, and the direct material damage has been estimated to be approximately 300 billion US dollars [4,5]. Given that 40% of world population lives within 100 km of coast and nearly 39% of USA population lives in counties directly next to coastlines [6], plus numerous associated coastal infrastructure, surges and waves have become a very serious threat to safety and intactness of coastal communities.

Owing to the importance of the problem, a number of investigations have been made on the impact of surges and waves on bridges. As early as the 1960s, hydrodynamic processes of solitary waves impinging two-dimensional plates were studied experimentally by El Ghamry [7] and French [8]. Since the disaster caused by Hurricane Katrina in 2005, research on the mechanism of bridge damage during extreme surges and waves has attracted significantly more attention. Various experiments have been carried out to study the hydrodynamic characteristics of bridge deck impacted by different types of waves [9–12]. An “ad-hoc” prediction method for both quasi-static and impulsive wave load based on experimental results was proposed by Cuomo et al. [9]. In addition to experimental

---

* Correspondence to: Hansong Tang, Department of Civil Engineering, City College, The City University of New York, 138th Street and Convent Avenue, New York, NY, 10031, USA.
E-mail address: htang@ccny.cuny.edu (H.S. Tang).

https://doi.org/10.1016/j.apor.2018.02.020
0141-1187/

work, different analytical methods have been proposed to predict hydrodynamic load on bridge decks [13–15], and various numerical models have been applied to study bridge damage mechanism using solitary waves [16,17], regular waves [18–20], and tsunami waves [21,22]. A research group carried out both experimental and numerical studies on the impact at bridge deck by solitary waves [23–25] and cnoidal waves [26], and variations of hydrodynamic wave loading under different wave heights and submergence depths were discussed in detail. Based on both experimental work [9,25] and numerical work [17,27,28], it has been proposed to install air vents at bridge decks to reduce the uplift component of the hydrodynamic force. However, it was founded that air vents might not always reduce hydrodynamic load; when a deck is initially submerged in water, the uplift and horizontal forces can be substantially higher than those without air vents [17]. Service roads were also proposed as an alternative measure to mitigate the risk due to excessive hydrodynamic load on bridges during surge events [29,30]. A more detailed review on past relevant work can be found in [31].

Ocean waves and flow currents co-exist in a natural marine environment, especially in the early stage of a storm surge event, when the water level is rising and a current is formed [32]. Their interaction has a significant influence on damage process of coastal bridges. However, nearly all previous studies on waves impinging bridges were carried out without consideration of currents and such interaction. In the present study, waves and currents as well are considered, and a numerical study will be made to examine the hydrodynamic characteristics of a bridge deck exposed to both of them. The influence of primary factors including current speed, submersion depth, wave height, and water depth has been discussed. Also, the effects of current on the efficiency of air vents at a bridge deck for reducing the hydrodynamic load have been studied. The rest of the paper is organized as follows. Section 2 presents the governing equations and corresponding numerical methods. Model calibration is described in Section 3. Section 4 presents the results and discussions in detail. Concluding remarks are provided in Section 5.

2. Numerical model

2.1. Governing equations

An unsteady, incompressible, two-phase flow is described by the three-dimensional Navier-Stokes equations, which read as

$$\nabla \cdot \boldsymbol{u} = 0 \tag{1}$$

$$\frac{\partial \rho \boldsymbol{u}}{\partial t} + \nabla \cdot (\rho \mathbf{u}\mathbf{u}) - \nabla \cdot (\mu_{\mathrm{eff}} \nabla \boldsymbol{u}) = \nabla \boldsymbol{u} \cdot \nabla \mu_{\mathrm{eff}} - \nabla p + (\rho - \rho_{\mathrm{ref}}) \cdot \boldsymbol{g} \tag{2}$$

where $\mathbf{u}$ is the flow velocity, t is the time, p is the pressure, ρ is the density of air-water mixture. ρ_{ref} is a reference density, $\mu_{\mathrm{eff}} = \mu_l + \mu_t$, being the effective viscosity including the laminar viscosity, μ_l, and turbulence viscosity, μ_t, and $\mathbf{g}$ is the acceleration of gravity. The reference density is used to eliminate the hydrostatic pressure accumulation in the gas phase region. The total pressure $p^* = p + \rho_{\mathrm{ref}} \mathbf{r} \cdot \mathbf{g}$, where $\mathbf{r}$ is the vector that starts from a reference point to the location where the total pressure value is calculated.

In the present model, the turbulent viscosity is determined using the Smagorinsky model [33], by which it is calculated as

$$\mu_t = \rho (C_s \Delta)^2 |\boldsymbol{e}| \tag{3}$$

where C_s is a coefficient between 0.1 and 0.2 ($C_s = 0.2$ is used in the present paper). $\mathbf{e}$ is the filtered strain rate tensor, and $\Delta = V^{1/3}$, with V being a control cell's volume.

The free surface between water and air is resolved using a volume of fluid method (VOF), which is originally proposed by Hirt and Nichols [34]. The transportation equation of volume of fluid has the following form

$$\frac{\partial \gamma}{\partial t} + \nabla \cdot (\gamma \boldsymbol{u}) = 0 \tag{4}$$

where γ is the volume fraction of water, and it is defined as

$$\begin{cases} \gamma = 0, & \textit{air} \\ 0 < \gamma < 1, & \textit{interface} \\ \gamma = 1, & \textit{water} \end{cases} \tag{5}$$

The local density and laminar viscosity as functions of are determined as

$$\rho = \rho_{\mathrm{air}} + \gamma \cdot (\rho_{\mathrm{water}} - \rho_{\mathrm{air}}) \tag{6}$$

$$\mu_l = \mu_{\mathrm{air}} + \gamma \cdot (\mu_{\mathrm{water}} - \mu_{\mathrm{air}}) \tag{7}$$

2.2. Computational method

In this paper, waves and currents are simulated using the Solver for Incompressible Flow on Unstructured Mesh (SIFUM), which we recently developed to solve the governing equations in above section [35]. In SIFUM, a computational domain is discretized by an unstructured hexahedral mesh to fit the complex solid boundaries. To adopt a high order advection scheme, the deferred-correction method proposed by Fergizer and Peric [36] is applied to discretize the convective term, and it is a combination of a first-order upwind scheme and the second-order Gamma scheme [37]. Additionally, as in [36], the diffusion term is approximated using central difference with values at auxiliary nodes which lie at the intersection of the cell face normal and straight lines connecting the node of the control volume and the neighboring nodes. The pressure gradient term is treated in the same way as the diffusion term. In addition, the gravity term is included as a source term. To compute the velocity and pressure with the discretized momentum equation, the momentum interpolation method proposed by Rhie and Chow [38] is applied to interpolate the velocities at the faces of the control cell. The continuity equation is integrated by summation of the volume fluxes across the faces of each control cell, and velocity and pressure are coupled using Pressure Implicit Split Operator (PISO) method proposed by Issa [39].

The convection equation of the volume fraction (4) is calculated using an algebraic VOF method. With this method, a finite volume method is applied to the convection equation, and the volume fraction at the cell face is interpolated with the Switching Technique for Advection and Capturing of Surfaces (STACS) from Darwish et al. [40]. STACS is a high resolution scheme based on the Normalized Variable Diagram (NVD) concept [41], which switches between different high-resolution differencing schemes to yield a bounded volume of fluid field according to free surface orientation. For temporal discretization, the Crank-Nicolson differencing scheme is used to calculate the next time step values by solving the discretized equation. It is generally recognized that predictions obtained by STACS are accurate with little diffusion.

In SIFUM, an adaptive time step has been determined from stability requirements by the gravity, convective, and viscous terms [42]. The convective time step is restricted by a CFL (Courant–Friedrichs–Lewy)-type constraint [43]

$$(\Delta t)_{\mathrm{conv}} \le \min \left(\frac{\Delta}{|\boldsymbol{u}|} \right) \tag{8}$$

The viscous time step restriction is given by

$$(\Delta t)_{\mathrm{visc}} \le \min \left(\frac{\Delta^2 \rho}{\mu_{\mathrm{eff}}} \right) \tag{9}$$