Finite Elements in Analysis and Design 47 (2011) 543–558

Contents lists available at ScienceDirect

Finite Elements in Analysis and Design

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

Finite element simulation of acoustic cavitation in the reservoir and effects on dynamic response of concrete dams

Farhoud Kalateh *, Reza Attarnejad

School of Civil Engineering, University College of Engineering, University of Tehran, P.O. Box 11365-4563, Tehran, Iran

ARTICLE INFO

Article history:
Received 13 June 2009
Received in revised form
3 December 2010
Accepted 10 December 2010

Keywords:
Fluid–structure interaction
Finite element method
Nonlinear analysis
Cavitation
Concrete dams

ABSTRACT

A 3D finite element formulation for the dynamic analysis of concrete dams is presented. A mixed Eulerian–Lagrangian formulation is used to simulate fluid–structure system. During severe ground motion, the impounding fluid in the reservoir may separate from the dam and causes forming of micro bubbles. As a result, the compressibility of water is reduced. This nonlinear phenomenon of the reservoir is termed as cavitation. When the direction of ground motion is changed, the micro bubble's region of fluid collapses, and an impact will occur. In order to eliminate the spurious oscillations, which are caused by the impact, a small amount of artificial stiffness proportional damping is added in the fluid domain. To capture cavitation effects a bilinear equation of state is employed and incorporated with finite element formulation of fluid domain. An iterative partitioned method is used to simultaneous time integration of equations of motion of fluid and structure domains. The developed method is validated by testing it against problem for which, there is existing solution. Also the effects of cavitation on dynamic response of Koyna gravity dam and Morrow Point arch dam subjected to the first 6 second of the May 1940 El-Centro, California earthquake, is considered. In order that truly consider the effects of cavitation phenomenon, maximum acceleration was scaled to give an amplitude of 1*g*. Obtained results show that impact force caused by cavitation has a small effect on the dynamic response of dam–reservoir systems.

1. Introduction

The formation and collapse of gaseous regions near the reservoir and dam surface due to cavitation is one of the sources of nonlinear behavior in the dynamic response analysis of dam–reservoir systems that may occur during intense seismic excitation. The collapsing of gaseous regions in the water alters the hydrodynamic pressure acting along the upstream face of the dam and affects its dynamic response. Cavitation is produced due to the pressure wave propagating in the reservoir medium interacting with the dam's structure and reflecting form dam's wet face and the free surface. It is manifested by the separation of the fluid medium from the structure at the interface. Such separation, i.e., cavitation, takes place because the structure–fluid medium interface is unable to transmit a specific change in the pressure wave intensity. For example, if a pressure wave, generated by an earthquake, encounters a structure, a gap, (i.e. cavitation) may occur between the fluid and an interface normal to the direction of the compressive pressure wave propagation. In this case, the fluid–structure interaction result in a (tensile) excessive force at the interface that is not tolerated by the fluid and, therefore, a gap is produced. Cavitations continue until the gap closes and linear fluid–structure interaction is resumed. Due to the complex nature of cavitation, this phenomenon is not an extensively discussed topic in the dam–reservoir interaction. It seems essential to consider the possibility of acoustic cavitation phenomenon for a realistic vibration analysis of the dam–reservoir systems. The dynamical behavior of coupled systems is characterized by different properties of the interacting subsystems that often describe different physical effects in the systems. Although, high-performance algorithms have been developed for many special models, it is often inefficient to apply these algorithms to the entire systems. Therefore, in this paper attention is focused on the development of an innovative numerical procedure for computing the cavitating fluid effects on the dynamic coupled response of the concrete dams. Although the main reason is to analysis the seismic response of dams, the proposed method is applicable to other fluid–structure systems in which the fluid is inviscid and undergoes small amplitude motions.

The possibility of acoustic cavitation forming in the reservoir has been shown analytically and observed in model tests [1]. Clough and Chang [2] have made an analytical study of cavitation for a gravity dam assuming incompressible water. They showed that impact of the water resulting from collapse of the cavitation bubbles could increase tensile stresses in the upper part of the dam by 20–40 percent. A more accurate representation of cavitation

* Corresponding author. Tel.: +98 411 3818046; fax: +98 914 415 4062.
E-mail addresses: f.kalateh@gmail.com (F. Kalateh), attarnjd@ut.ac.ir (R. Attarnejad).

0168-874X/$ - see front matter © 2010 Elsevier B.V. All rights reserved.
doi:10.1016/j.finel.2010.12.004

requires consideration of fluid compressibility, such as the bilinear constitutive model for cavitating inviscid fluid proposed by Bleich and Sandler [3] that presents a numerical treatment of cavitating fluids similar to smeared crack approach related to the modeling of cracking propagation. Zienkiewicz et al. [4] used this bilinear fluid model to study the concrete dams and to clarify the phenomenon of cavitation. They distinguished between the cavitation due to fluid flow and that due to elastic wave propagation. In the former, cavitation occurs due to the high flow velocity, which in turn reduces the absolute fluid pressure to below zero. This results in a periodic formation and the subsequent collapse of vapor bubbles in the high velocity region of the fluid domain. In the latter case, cavitation is attributed to the fluid expansion and often results in isolated regions of cavitation. The authors concluded that cavitation would not alter the maximum stresses of concrete gravity dam significantly. Rose et al. [5] presented a new method to couple the fluid and solid fields, based on the localized Lagrange multiplier (LLM) for studying the interaction of an acoustic fluid with flexible structure and used a displacement potential formulation for modeling the cavitating fluid.

Due to these limited studies, the importance of cavitation in the earthquake response of concrete dams is still ambiguous. Specially, there has been no study of the cavitation effects on the concrete arch dams, where the dam–water interaction effects are more important than for the gravity dams [6]. In the literature, there are considerable disagreements on the importance of modeling the cavitation formation in the reservoir in the evaluating of the seismic response of concrete dams.

Various numerical methods have been employed to simulate the cavitation phenomenon in the liquid using finite element method (a detailed description can be found in Refs. [7,8]). These methods can be categorized into four main classes: (1) using pressure based finite element formulation (FEM) for the fluid domain and the displacement-base finite element for the structure domain [9]. (2) Applying displacement base FEM for both fluid and structure domains [10,12,13]. (3) Formulate the fluid response in terms of a potential function for displacement or velocity incorporating with the displacement base FEM for the structure domain [4,5,7,13,14]. (4) Another approach for formulating fluid domain involves a combination of the formulations mentioned above that is usually named hybrid formulation of fluid domain [8,15].

A major problem that arises during modeling inviscid fluids using displacement based Lagrangian formulation is that due to the absence of shear stresses, spurious zero energy modes may contaminate the numerical results. Such problems have been reported in the past by various researchers. To eliminate zero energy modes, several stabilization methods have been proposed [16], such as the introducing of artificial stiffness and the enhanced strain method. However, assuming the pressure as independent variable to represent the fluid motion has certain advantages in this context. First, the compressibility of the liquid is considered in a natural way and does not increase the computational effort and cost significantly. Second, the additional computational step of finding pressure, inherent in the potential based formulation, is unnecessary. This can save considerable amount of computational time depending on the problem size and time integration technique employed [17].

From the literature review carried out above, the way of modeling the cavitation phenomenon at the fluid–structure interfaces seems to be an important and promising subject. The question of whether the cavitation is important for simulating the realistic response of arch dams will be addressed in this study. Therefore, a special program has been developed for calculating the modified hydrodynamic pressure distribution along upstream face of dam, with considering the reservoir cavitation. In the proposed method, mixed Eulerian–Lagrangian finite element formulation has been used, for modeling the fluid and solid domains, respectively. Following the work of Sandberg [15], a modified method has been developed for modeling the cavitation effects in reservoir based on improvement the stiffness matrix of the cavitated fluid finite element, developed scheme is implemented into a substructure model configuration of fluid–solid interaction. A stabilized iterative partitioned method is utilized to solve the equation of motions of involved fields. Applying this solution technique, the equation of motion of each field separately is solved at each time step and interaction is forced through their interface. Therefore, assembling matrices with a large bandwidth is not required. This saves the required memory, especially for the cases with non-symmetric matrices. In the present work, the equation of motion governing the fluid is expressed in terms of pressure alone considering the fluid as compressible, inviscid and irrotational. Comparative studies clearly show the efficiency and effectiveness of the proposed method. The system under consideration, which is shown in Fig. 1, consists of a structure domain and a fluid domain with their common interface and other five surface boundaries.

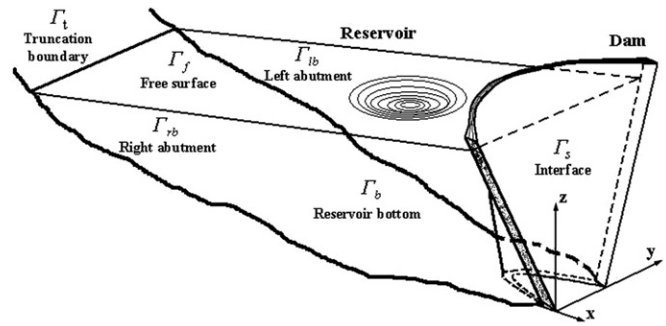

Fig. 1. Dam–reservoir system and definition of reservoir boundaries.

2. Governing equations for the fluid medium

The general governing equations for a compressible and viscous fluid can be written as

Continuity equation

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho_0 u_i)}{\partial x_i} = 0 \tag{1}$$

Momentum equation

$$\frac{\partial(\rho_{f0} u_i)}{\partial t} + \frac{\partial(\rho_{f0} u_i u_j)}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} \tag{2}$$

$$\tau_{ij} = \mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \tag{3}$$

and the equation of state (EOS)

$$P = f(\rho, \dot{\rho}) = \alpha(\rho)\rho + \beta(\rho)\frac{\partial \rho}{\partial t} \tag{4}$$

In the present work, the momentum equation is simplified to represent an inviscid fluid that undergoes small amplitude and irrotational motion, therefore we can write

$$\rho_{f0}\frac{\partial u_i}{\partial t} + \frac{\partial P}{\partial x_i} = 0 \quad or \quad \rho_{f0}\frac{\partial u_i}{\partial t} + \nabla P = 0 \tag{5}$$

where ρ, P, u_i, μ and τ_{ij} are density, pressure, velocity, viscosity and the viscous stress tensor of the fluid, respectively. x_i and t denote the spatial coordinates in the ith direction, and time, respectively. ρ_{f0} is the reference value of the fluid density and f is an arbitrary, nonlinear function of ρ and $\partial\rho/\partial t$. The nonlinear convection term $\partial(\rho_0 u_i u_j)/\partial x_j$ in Eq. (2) can be neglected for the acoustic fluid when the fluid velocity is small compared to the dimensions of the model.