Numerical Modeling and Experimental Visualization of Wave Propagation over Semicircular Obstacles

Tamer H. M. A. Kasem and Jun Sasaki
Department of Civil Engineering, Yokohama National University
Yokohama, Japan

ABSTRACT

Periodic wave propagation over a semicircular obstacle is studied. To provide accurate modeling of separated flow viscous flow is adopted in the current numerical model. Convection terms are discretized using the fifth order space accurate WENO method. Experimental visualization is also performed. Well agreement between both calculated and visualized free surface profiles is obtained. The numerical results reveal the existence of flow vortices in both the water and the air phases. Drag force in the negative direction is found to be larger than that in the positive wave propagation direction. The model results should provide important guidance for engineers.

KEY WORDS: Semicircular Breakwater; Drag Force; WENO5; CFD; Visualization.

INTRODUCTION

Detached or nearshore breakwaters have been used extensively for coast protection or the creation of crescentic beaches (Reeve et al., 2004). However in order to achieve some understanding of the complex fluid motion associated with wave-structure interaction, it is necessary to simplify the geometry of coastal structures (Chang et al., 2001). For example a submerged rectangular box has been considered as a simplified model for a submerged breakwater in many research papers listed by Chang et al. (2001). Following the same principle, a semicircular obstacle which is the topic of the present work may be used to model a semicircular breakwater. A semicircular breakwater is a relatively new type of structures which were first developed in 1990s. Semicircular breakwaters are expected to exhibit superb characteristics in terms of stability (Sasajima et al., 1994). Upon reviewing the current literature, the need for more studies of the problem of wave propagation over semicircular breakwaters is observed. This can be attributed to the limitations found in the studies published so far. The studies dealing with this topic may be generally classified into two types: a) numerical studies and b) experimental and field works.

One example of the few numerical studies is the work of Yuan and Tao (2003). In this study elaborate results were presented, however a potential inviscid flow model was adopted. Zhuang and Lee (1996) verified using experiments and calculations that a major drawback of potential flow assumption is that it may lead to large errors for separated flow with vortices behind an obstacle. Although Zhuang and Lee (1996) considered only the case of a rectangular obstacle, the same drawback is expected for a semicircular shape. In addition Chang et al. (2001) pointed out that these localized rotational and dissipative vortices have a definite impact on the mixing process, sediment transport and scouring process. Viscous flow is adopted in the current numerical model to provide adequate simulation of flow separation and vortices. Convection terms are discretized using the fifth order space accurate WENO method. Upon using such high order method the disadvantages of low order methods used in COBRAS (Lin and Liu, 1998), VOFbreak2 (Troch and De-Rouk, 1998), and CADMAS-SURF (Isebe et al., 1999) models are avoided. The two main disadvantages of the low order methods are the excessive diffusion of vortices, and the need of grid clustering (Ekaterinaris, 2005).

The works of Sundar and Ragu (1998) and Sasajima et al. (1994) are two examples of experimental and field studies, respectively. In both works extensive results of pressure measurement were presented. However free surface visualization data were totally missing. These data are useful for two reasons. Firstly visualization of flow near the obstacle provides more understanding of the problem dynamics. Also these data may be used to test and validate the current two-phase numerical model and future models developed by other researchers. An important goal for this work is providing reliable experimental visualization data using inexpensive equipments.

NUMERICAL MODEL

In the current section the numerical model used is presented, and the model main remarkable features are stressed. However due to space limitation, a detailed description of the model including validation against various test cases will be published in a separate work.

Governing Equations

Neglecting surface tension for a two-phase gas-liquid flow, the