International Journal of Offshore and Polar Engineering (ISSN 1053-5381)

Vol. 15, No. 3, September 2005, pp. 198–203

Numerical Model of 2-D Multiphase Flow with Solid-Liquid-Gas Interaction

Koji Kawasaki*
Department of Civil Engineering, Nagoya University, Nagoya, Japan

In this study, a 2-dimensional numerical model of multiphase flow is developed based on a CIP method and an extended SMAC method. In order to validate the proposed numerical model, hydraulic phenomena with not only gas-liquid interfaces but also solid-gas-liquid interfaces are computed; for example, collapse of a liquid column, gravity current surge due to exchange of 2 fluids, collision between a collapsing water column and a solid body, and free fall of a rigid body onto a still water surface. It is demonstrated from the computational results that the numerical model can provide stable results for both gas-liquid phase flow and solid-gas-liquid phase flow, and simulate the hydraulic phenomena precisely.

INTRODUCTION

Understanding the mechanism of air-water interaction is very important from the viewpoint of fluid dynamics in estuaries, lakes and coastal sea areas. Wind-induced upwelling of anoxic bottom water in stratified estuaries, and wave breaking with air bubble entrainment are two examples of such physical phenomena.

Recently, numerical attempts to clarify the internal structures of complicated physical phenomena have been actively made through the development of high-speed calculation processors and advanced numerical schemes. Kawasaki and Iwata (1998) and Kawasaki (1999) have studied wave-breaking and post-breaking wave deformation due to a submerged breakwater in the 2- and 3-dimensional wave fields using a VOF (Volume Of Fluid) method developed by Hirt and Nichols (1981). Yabe and Wang (1991) developed a C-CUP (Cubic interpolated propagation–Combined Unified Procedure) method to solve compressible and incompressible fluid dynamics simultaneously. Sussman et al. (1994) proposed a level set method, which can compute the interface between 2 fluids by introducing a distance function from the interface. However, in general it is difficult to simulate the multiphase flow for the incompressible and compressible fluids simultaneously, because the physical properties of each phase, such as density, viscosity coefficient and so on, are quite different.

The main purpose of this study is to develop a 2-D numerical model of multiphase flow that includes solid, liquid and gas phases based on a CIP (Cubic Interpolated Propagation) method and an extended SMAC (Simplified Marker And Cell) method. The model is verified by applying it to some hydraulic phenomena such as collapse of a liquid column, gravity current surge, collision between a collapsing water column and a solid body, and free fall of a rigid body onto a still water surface.

2-D MULTIPHASE FLOW MODEL

Governing Equations

The governing equations consist of the conservation equation of mass, the Navier-Stokes equations in the respective directions of x and z, the pressure equation for compressible fluid, the conservation equation of density function, and the equation of state for barotropic fluid, as shown in Eqs. 1~6:

$$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho w)}{\partial z}=0 \tag{1}$$

$$\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+w\frac{\partial u}{\partial z}=-\frac{1}{\rho}\frac{\partial p}{\partial x}+F_x \tag{2}$$

$$\frac{\partial w}{\partial t}+u\frac{\partial w}{\partial x}+w\frac{\partial w}{\partial z}=-\frac{1}{\rho}\frac{\partial p}{\partial z}+F_z \tag{3}$$

$$\frac{\partial p}{\partial t}+u\frac{\partial p}{\partial x}+w\frac{\partial p}{\partial z}=-\rho C_s^2\left(\frac{\partial u}{\partial x}+\frac{\partial w}{\partial z}\right) \tag{4}$$

$$\frac{\partial \phi_I}{\partial t}+\frac{\partial(\phi_I u)}{\partial x}+\frac{\partial(\phi_I w)}{\partial z}=0 \tag{5}$$

$$\rho=f(p) \tag{6}$$

where ρ is fluid density, u and w are velocity components in the respective directions of x and z, p is pressure, F_x and F_z are external force terms such as viscous term, gravitational force term and surface tension term, C_s is local sound speed, and t is time. ϕ_I ($I=1\sim3$) are density functions for respective phases (ϕ_1: solid phase; ϕ_2: liquid phase; ϕ_3: gas phase) that represent the rate of fractional volume for each phase in a cell. These functions need to satisfy the relationship of $\phi_1+\phi_2+\phi_3=1$ ($0\le\phi_1, \phi_2, \phi_3\le1$).

Computational Algorithm

Fig. 1 shows the computational algorithm of the multiphase flow model. Governing Eqs. 1~5 are separated into an advection step and a nonadvection step using a time-splitting method, as indicated by Eqs. 7~16. The CIP method developed by Yabe and Aoki (1991) is used to calculate the hyperbolic equations for all variables at the advection step, while the equations at the nonadvection step are solved with an extended SMAC method, which is derived in this study so as to simulate both compressible and incompressible fluid flows. The numerical procedure of the extended SMAC method is described later in detail. The effect of surface tension on the gas-liquid interface is evaluated by using a CSF (Continuum Surface Force) model proposed by Brackbill et al. (1992), which interprets surface tension as a continuous mass force across the interface. In computations, the time step interval has to satisfy the CFL (Courant-Friedrichs-Levy) condition, in which the Courant number is less than 1.

Advection step:

$$\frac{\partial \rho}{\partial t}+u\frac{\partial \rho}{\partial x}+w\frac{\partial \rho}{\partial z}=0 \tag{7}$$

*ISOPE Member.

Received October 8, 2004; revised manuscript received by the editors April 2, 2005. The original version was submitted directly to the Journal.
KEY WORDS: Multiphase flow, gas-liquid interaction, solid-gas-liquid interaction, numerical model, CIP method, extended SMAC method.