(Abstract for the 9th International Workshop on Water Waves & Floating Bodies, April 1994)

Nonlinear Waves at the Interface of Water and Mud in a Dredged Channel

J.W. Kim and K.J. Bai
Department of Naval Architecture and Ocean Engineering
Seoul National University, Seoul, Korea

Introduction

In this paper we simulate a two-dimensional internal wave generated by a moving disturbance. The present study is motivated by earlier experimental works of Ferdinande and Vantorre (1991) and Vantorre (1991) (hereafter referred as FV). In their papers, self-propulsion test is performed for a dredger ships of small keel clearance with muddy bottom. The speed of the ship so slow that the water surface is almost calm but the interfacial waves are developed and cause anomalies in steering. They have also performed an analysis using a simplified theory based on mass conservation and Bernoulli's equation. They discussed that a steady hydraulic jump can be exist under the ship. But it can be easily shown that the steady hydraulic jump of this kind cannot be exist by considering the conservation of momentum along the parallel middle body of the ship.

In the present study, the experiment of FV is numerically simulated using a finite-element method. The three-dimensional experimental model is replace by a two dimensional model where the ship is replaced by a two-dimensional body with the same sectional blockage coefficient. In the computations we could not obtain a steady state result. But the computed interfacial elevation taken at a selected time step shows a good agreement with the experimental result of FV.

Mathematical modeling

The previous experiment of FV, the configuration shown in Fig.1, is mathematically modeled as follows.

In the mathematical modeling of the present problem, only the effects of the inertia and the gravity are taken into account. The viscosity and the interfacial tension are neglected. Under these assumptions there are two major parameters which characterize the behaviour of the free surface for a given depth ratio h_2/h_1: The one is the Froude number $F_1\left(=U/\sqrt{gh_1}\right)$, which determines the characteristics of the air-water interface, and the other is the densimetric Froude number F_2, which determines the characteristics of the water-mud interface and is defined as

$$F_2 = U \Big/ \sqrt{\frac{g(\rho_2-\rho_1)h_1 h_2}{\rho_2 h_1 + \rho_1 h_2}} \tag{1}$$

A practical range of the Froude numbers is given as

$$F_1 \ll 1, \quad F_2 = O(1). \tag{2}$$

Under this circumstance the deformation of the upper free surface is negligibly small. Hereafter we will consider only the wave elevation at the water-mud interface. A further simplification is made by replacing the original three-dimensional problem by a two-dimensional model as shown in Fig.2. The channel is replaced by two dimensional wave flume with the same water- and mud-depths. The ship is replaced by a two dimensional object with the same sectional blockage coefficient S(x), defined as

$$S(x) = A(x) / Wh_1, \qquad (3)$$

where A(x) is the cross-section area of the ship.

Method of solution

With the assumptions given above, the present problem can be formulated as a partial differential equation in terms of the surface elevation $\zeta(x,t)$ and the velocity fields u1(x,y,t) and u2(x,y,t) defined in the water and mud region, respectively. For the numerical computations based on the finite-element method, it is more convenient to formulate the present problem in a variational form. Hamilton's principle is used here. In this formulation the equation of motion is replaced by the stationary condition of a functional $J = \iint \mathcal{L}\, dxdt$, where the Lagrangian density $\mathcal{L}$ for the present problem can be written as

$$\mathcal{L} = \frac{\rho_1}{2}\int_{\varsigma}^{h_1} \mathbf{u}_1 \cdot \mathbf{u}_1 \, dz + \frac{\rho_2}{2}\int_{-h_2}^{\varsigma} \mathbf{u}_2 \cdot \mathbf{u}_2 \, dz - \frac{g(\rho_2 - \rho_1)}{2}\varsigma^2 + \Delta\phi\left(\varsigma_t - \frac{\mathbf{u}\cdot\mathbf{n}}{n_z}\right)_{z=\varsigma} \qquad (4)$$

where $\mathbf{n}=(n_x,n_z)$ denotes the unit normal vector on the interface. The first two terms are kinetic energies of the fluid. The next term is the potential energy. The new variable $\Delta\phi$ in the last term is a Lagrange multiplier for the kinematic interface boundary condition which can be shown to be equivalent to the potential difference on the interface, i.e.

$$\Delta\phi = (\rho_1\phi_1 - \rho_2\phi_2)_{z=\varsigma} \qquad (5)$$

The trial solutions for the velocity fields are assumed to satisfy the kinematic constraints such as continuity equation in fluid domain and the kinematic boundary conditions on the boundaries. The trial solution can be constructed by adopting stream function $\psi(x,z,t)$ which is represented as

$$\psi_1(x,z,t) = \sum_{m,n} Z_m(\gamma_1) X_n(x) \psi_{1mn}(t), \quad \psi_2(x,z,t) = \sum_{m,n} Z_m(\gamma_2) X_n(x) \psi_{2mn}(t) \qquad (6)$$

where $X_n(x)$ and $Z_m(\gamma)$ are the finite-element interpolation functions in x and z direction, respectively. Here, the transformed coordinate γ_1 and γ_2 are defined as

$$\gamma_1 = \frac{z-\varsigma}{H_1-\varsigma}, \quad \gamma_2 = \frac{z-H_2}{\varsigma-H_2} \qquad (7)$$

where H_1 and H_2 denote the z-coordinate of the upper and the lower solid surfaces, respectively. By substituting Eq.(6) into Eq.(4), we obtain the Larangian in terms of the discretized stream function

$\{\psi_{1mn}(t), \psi_{2mn}(t) | m = 1,\ldots,M; n = 1,\ldots,N\}$. Then, from the Euler-Lagrange equation, we can obtain ordinary differential equations of the discretized stream functions, which can be easily integrate numerically. Details of the numerical scheme can be found in Bai et al (1989) and Choi et al (1990).

Numerical results and discussion

As a computational model, one of the experimental condition of FV is taken. The principal dimensions and the test conditions are given as follows

$$L = 2.888\ \text{m},\ B = 0.575\ \text{m},\ T = 0.200\ \text{m},\ W = 2.25\ \text{m},$$
$$h1 = 0.240\ \text{m},\ h2 = 0.035\ \text{m},\ \rho2/\rho1 = 1.11$$

The computed result with U = 0.136 m/s is presented here. In this case the computed Froude numbers are

$$F_1 = 0.09, \quad F_2 = 0.78. \tag{8}$$

In Fig.3 the wave profiles at several time steps are plotted. The interfacial elevation shows depression at the bow and uprising at the stern in initial stage. After then the elevation forms a structure like a undular bore underneath the ship. The front of the bore propagates to upstream with speed faster than U. In the theoretical analysis of FV, a steady hydraulic jump under the ship is predicted. However in the present computed results, an unsteady undular bore is observed. In Fig.4 the computed wave profile is compared with the experimental result taken from FV. Since they did not make it clear whether the surface elevation is in steady state and did not give the location of the measurement explicitly, we selected the time step such that the location of the computed wave-front can be fitted to the measured data. We can find that the two results agree qualitatively well. The two straight lines indicate the up- and down-stream elevations of the hydraulic jump predicted by FV.

As a concluding remark, it should be noted that the interfacial layer was unstable throughout the computation. This is presumably due to the Kelvin-Helmholtz instability caused by discontinuity of tangential velocity across the interface. To stabilize the numerical solution, a diffusion model with an artificial damping term is introduced in the dynamic interface condition. Presumably, the artificial damping plays a role of stabilizer as the viscous does in the real situation. For a more realistic simulation, a more reasonable diffusion model should be adopted based on the theoretical or empirical consideration of the viscous effects.

References

Ferdinande, V. and Vantorre, M. 1991 "The behaviour of a mud-water interface underneath a slowly advancing ship at small keel clearance," *Proc. Int. Sympo. on Hydro- and Aerodynamics in Marine Eng.*, Bulgaria, pp 4.1-4.10.

Vantorre, M. 1991 "Ship behaviour and control at low speed in layered fluids," *Proc. Int. Sympo. on Hydro- and Aerodynamics in Marine Eng.*, Bulgaria, pp 5.1-5.90.

Bai, K.J., Kim, J.W and Kim, Y.H. 1989 "Numerical Computations for a Nonlinear Free-Surface Flow Problem," *Proc. 5th Int. Conf. on Numerical Ship Hydrodynamics*, Hiroshima, Japan, pp 403-419

Choi, H.S., Bai, K.J., Kim, J.W. and Cho, I.H. 1990 "Nonlinear Free Surface Waves Due to a Ship Moving Near the Critical Speed in a Shallow Water," *Proc. 18th Sympo. on Naval Hydrodynamics*, Univ. of Michigan, USA, pp 173-189

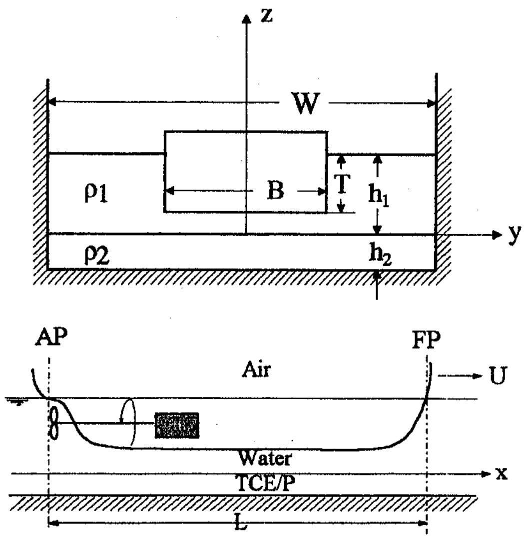

Fig.1 Configuration of physical model

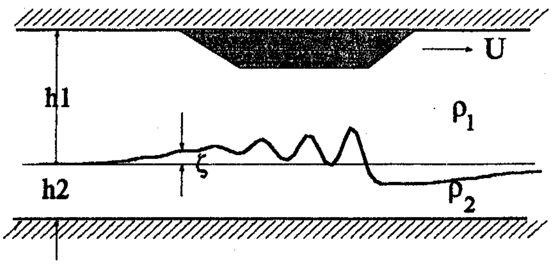

Fig.2 Configuration of two-dimensional theoretical model

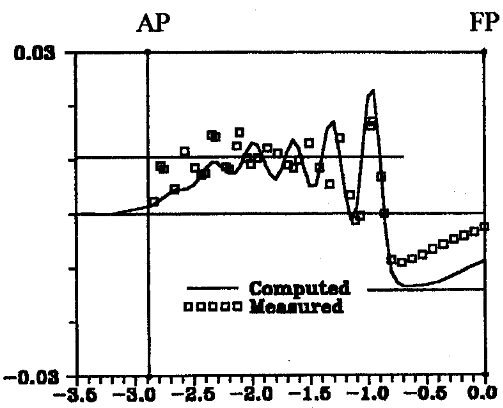

Fig.4 Computed and measured interfacial elevation

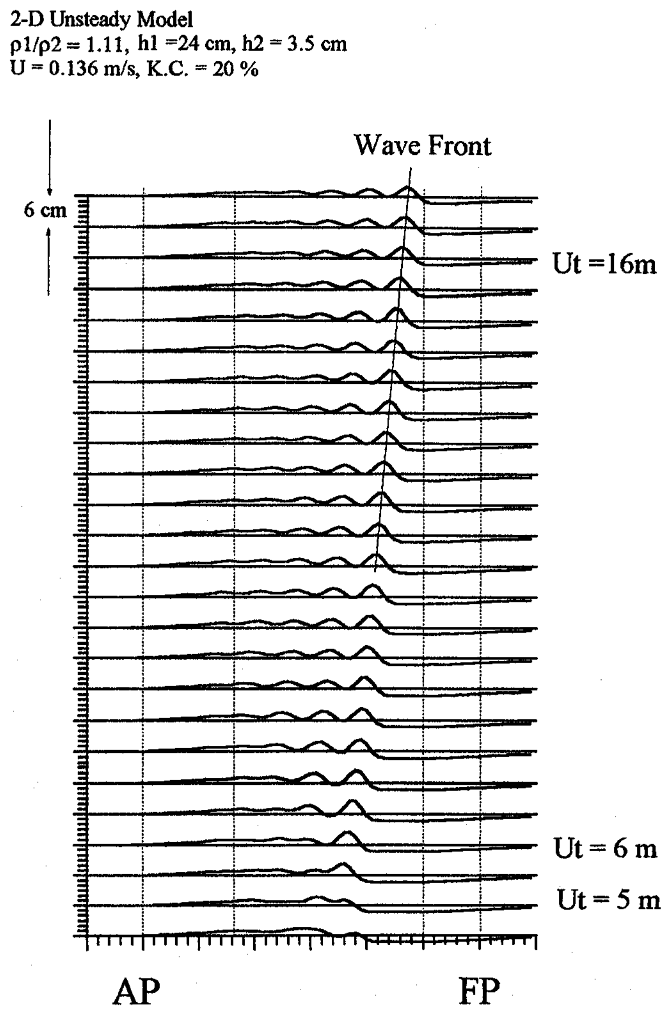

Fig.3 Time history of interfacial elevation