Numerical Modeling of Flow in Water Entry of a Wedge

Hajime KIHARA
National Defense Academy, Yokosuka, JAPAN
Email : hkihara@nda.ac.jp

Abstract

The water entry problems of a wedge with constant vertical velocity are studied. A boundary element method (BEM) and a mixed Euler and Lagrangian method (MEL) are employed for the analysis of the problems. The computational models about jet flow are proposed and these are applicable to the problems for wedges with various dead rise angles. Computational results of the hydrodynamic pressure on wedges show good agreements with other theoretical ones. In addition, it turned out that the singularities at a bottom vertex were apt to lose the precision of local pressure values. Accordingly we introduce the analytical description of the flow in a local domain around the vertex. These solutions can be determined by solving integral equations with the solutions in the other domain.

Introduction

Hydrodynamic impact analysis taking account of the local uprise of water was initiated by Wagner. Theoretical studies are extended up to the analysis including jet flow due to the water impact these days and they are still challenging issues. Jet generation is one of characteristic features in water impact phenomena. A thin jet layer is formed along the body surface in the case of water impact of a wedge. The analysis by matched asymptotic expansions was presented recently [2][5]. The BEM is also one of numerical procedures where a jet region can be taken into account. The time-stepping method is employed together to update moving boundaries generally. Zhao & Faltinsen [9] introduced the so-called "cut-off" model of jet flow. This model provides the good prediction of hydrodynamic pressure in the problems with various dead rise angles. However, as the jet flow is cut off at the spray root, we cannot obtain the information, for instance, the evolution of the jet flow and the separation from the body surface. Recently the interesting method that can make up with this drawback was presented by Iafrati [6]. The jet region is divided into several small panels and the velocity potential on each panel is computed by using local Taylor expansions and matching with the other domain.

The present work is intended to develop the numerical method by the BEM, which can describe the evolution of the jet flow in the water entry problem of a wedge. For this purpose, we introduce two computational models about the jet flow. One is the model to cut off the jet flow, and the other is the model about the convexity of the free surface shape. Particularly, the cut-off operation is restricted only to the jet tip in order to keep computed information. The idea may be conceptually similar to the method by Fontaine & Cointe [3], although their computational results near jet tips are different from the present ones. As we describe how

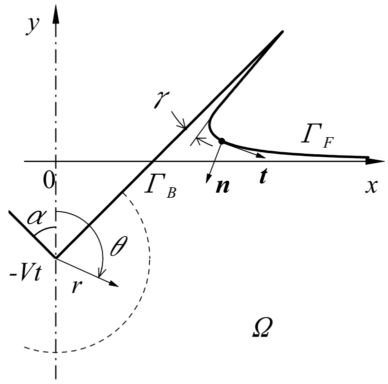

Fig. 1: Coordinate system.

to cut off the jet flow in the later section, we'd like to emphasize that such a manner is arbitrary to some extent. It suggests that the exact description of the flow is not always necessary within the jet region. The second computational model is convenient to expose the influence of gravity in the jet flow. These computational models enable the total simulation up to re-entry against the underlying free surface, e.g. an early stage, a self-similar stage, an deformation stage where gravity effects becomes dominant and a final stage leading to a plunging jet. Another motivation for this work is the flow domain around the vertex of a wedge [8] and attention is necessary to deal with numerical analysis. The presence of the singularities at the vertex also affect the computation of the hydrodynamic pressure locally. It turned out that the introduction of analytical forms for the velocity potential in this local domain brought some advantages. For instance, the computational errors in numerical differential due to both a type of its scheme and nodal intervals can be avoided. This is suitable to describe the body surface boundary condition particularly in the acceleration field.

Formulations of problems

The water entry problem of a symmetrical wedge moving down with constant velocity V is considered. The x and y axes are taken along the undisturbed free surface and along the body centerline pointing upward, respectively, as is shown in Fig.1. Assuming the fluid is incompressible and the flow is irrotational, the fluid motion is specified by the velocity potential ϕ. The problem is governed by the following equations:

$$\nabla^2 \phi = 0 \qquad \text{in} \quad \Omega \tag{1}$$

$$\frac{\partial \phi}{\partial n} = -V n_y \qquad \text{on} \quad \Gamma_B \tag{2}$$

$$\frac{D\phi}{Dt} = \frac{1}{2} |\nabla \phi|^2 - gy \qquad \text{on} \quad \Gamma_F \tag{3}$$

$$\frac{D\boldsymbol{x}}{Dt} = \nabla \phi \qquad \text{on} \quad \Gamma_F \tag{4}$$

where g and $\boldsymbol{x}$ denote the acceleration of gravity and the position vector of arbitrary point in the domain, respectively, and the other symbols are defined in Fig.1. Since the time derivative of the velocity potential is necessary to compute the hydrodynamic pressure, the boundary value problem about the acceleration field is also formulated in the present work as follows:

$$\nabla^2 \phi_t = 0 \qquad \text{in} \quad \Omega \tag{5}$$

$$\frac{\partial \phi_t}{\partial n} = \frac{\partial \phi}{\partial n}\frac{\partial^2 \phi}{\partial s^2} - \frac{\partial \phi}{\partial s}\frac{\partial}{\partial s}\frac{\partial \phi}{\partial n} \qquad \text{on} \quad \Gamma_B \tag{6}$$

$$\phi_t = -\frac{1}{2} |\nabla \phi|^2 - gy \qquad \text{on} \quad \Gamma_F \ . \tag{7}$$

The initial conditions of the free surface are necessary to complete the problems. The conditions are given by

$$\phi = \phi_t = 0 \qquad \text{on} \quad \Gamma_F, \quad \text{at} \quad t = 0 \tag{8}$$

Thus the water entry problems of a wedge are formulated as the initial value - boundary value problems.

Numerical procedures and results

The solution procedures of problems are based on the MEL method. The fourth-order Runge-Kutta method was employed as a time-marching scheme in the present work. On the other hand, two sets of boundary value problems in equations (1)-(4), (5)-(7) are solved using the BEM.

Solution procedure of integral equation

Applying Green's theorem to each boundary value problem, the integral equation with the same form is obtained as follows:

$$C(P)\begin{Bmatrix}\phi(P)\\ \phi_t(P)\end{Bmatrix} + \int_\Gamma \begin{Bmatrix}\phi(Q)\\ \phi_t(Q)\end{Bmatrix} \frac{\partial G(P,Q)}{\partial n_Q}\, d\Gamma(Q) = \int_\Gamma \begin{Bmatrix}\frac{\partial \phi(Q)}{\partial n}\\ \frac{\phi_t(Q)}{\partial n}\end{Bmatrix} G(P,Q)\, d\Gamma(Q) \tag{9}$$

where $G(P,Q)$ is the Green function and expressed as $\frac{1}{2\pi}\ln|\boldsymbol{x}(P)-\boldsymbol{x}(Q)|$, and $C(P)$ denotes the interior angle at the observation point P.

The linear isoparametric elements are used for the discretization of equation (9). As the analytical integration along each element is performed, accurate results are provided even in the thin area near the jet tip. Initially we introduced a double node at the vertex, but desirable results about hydrodynamic pressure at the vertex could be not obtained for the presence of singularities at the vertex [8]. Next, we made the computational point shifted inside by using a constant element or a non-conformity element as the element adjacent to the vertex. The improvement was recognized to some extent, but these are easy to make computational errors due to numerical differential. Therefore, to reduce such disadvantage, the analytical expression of velocity potential is adopted in the local domain around the vertex as follows:

$$\phi = V \sum_{m=1}^{M} a_m\, r^{\frac{(m-1)\pi}{\pi-\alpha}} \cos\left\{\frac{(m-1)(\theta-\alpha)\pi}{\pi-\alpha}\right\} - V\, r\cos\theta \tag{10}$$

where a_m is a coefficient to be determined by matching with the solutions in the other domain. Equation (10) satisfies both the Laplace equation and the body boundary condition.

When equation (9) is discretized by using $(N-1)$ elements on the boundary, the simultaneous equations on N unknowns are obtained as follows:

$$[H_{ij}]\{\phi_j\} = [G_{ij}]\{\phi_{nj}\} \qquad (\, i, j = 1, 2, \cdots, N\,) \tag{11}$$

The discretized form of equation (10) can be written

$$\{\phi_k\} = [F_{km}]\{a_m\} + \{b_k\} \qquad (\, k, m = 1, 2, \cdots, M < N\,) \tag{12}$$

The matrix and vector of right-hand side are given by

$$\begin{cases} F_{km} = V r_k^{\sigma_m} \cos\{\sigma_m(\theta-\alpha)\}\ , \\ b_k = V r_k \cos\theta\,, \\ \sigma_m = \dfrac{(m-1)\pi}{\pi-\alpha}\ . \end{cases} \tag{13}$$

Let the local boundary (M nodes) locate in the first half of the total boundary (N nodes). Substituting equation (13) to the left-hand side of (11), the matrix form is

$$\left[\begin{array}{c|c} H_{ik} & H_{il} \end{array}\right]\begin{Bmatrix}\phi_k\\ \hline \phi_l\end{Bmatrix} = \left[\begin{array}{c|c} H^*_{im} & H_{il} \end{array}\right]\begin{Bmatrix}a_m\\ \hline \phi_l\end{Bmatrix} + \begin{Bmatrix} b^*_i \end{Bmatrix} \tag{14}$$

The new matrix and vector with * can be computed by

$$[H^*_{im}] = [H_{ik}]\,[F_{km}] \tag{15}$$

$$\{b^*_i\} = [H_{ik}]\,\{b_k\} \tag{16}$$

The total number of unknowns is the same as that before introducing equation (10). Therefore, using equation (14), we can solve the simultaneous equations with unknown coefficients a_m matched with the velocity potential ϕ_l in the remaining domain. In the present work, the radius distance of the local domain is set to $Vt/2$. Since the tangential second derivative of equation (10) is infinity at the vertex, a non-conformity element only at that point is used together.

Computational model of jet flow

At the intersection between the body and the free surface, a double node is placed for computation. When the boundary value problem is solved, we can compute the velocity for the intersection particle by taking account of not only the pressure condition but also the kinematic condition including interactions with the body's motion. The adopted treatment of the intersection give reliable computational results [7] in large amplitude oscillating problems. In our approach, the intersection particle is always on the body surface if the curvature of the body surface is zero. As is pointed out by Greenhow [4], the present computation also gives small negative pressure in jet region during the early stage of impact. However, the negative pressure disappears more and more, as the computational stage moves to the self-similar one. We can suppose that this is partly because the potential flow is applied to the jet region in spite of more complex flow actually. But we cannot specify the reason of this fact.

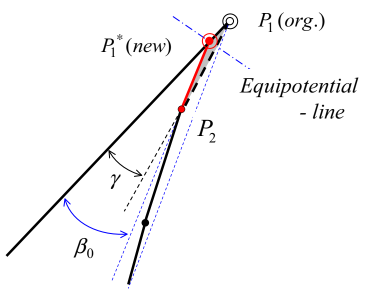

Fig. 2: Cut-off model of a jet tip.

In the case of abruptly starting motion like water entry of a wedge, so large acceleration is induced near the intersection and fluid particles run up along the body surface quickly. Since the singularities on velocity are in presence at the intersection, the numerical errors are inherent in the computation of the intersection for short duration after impact. Although we cannot conclude how much affects the sequential computation, the computation initiated by several different initial conditions arrives at the same state soon. We can trace the motion of the jet tip to some extent, but computational efforts will be added more and more because of the increasing computational domain and the numerical instabilities bringing the small negative pressure. As shown in Fig.2, the jet flow forms a thin triangle layer with an apex at the intersection. Since the velocity potential comes to have almost symmetry values on both boundaries, the flow can be considered one-dimensional toward the intersection. Generally speaking, a time step size need to be made enough small. This is helpful to prevent the free surface in the jet region from touching with the body surface. From the above-mentioned considerations, we introduce cut-off models to describe the jet flow practically. The tip shape of the jet region is controlled only by using a contact angle γ at the intersection. Its conceptual illustration is shown in Fig.2, where it is suggested that the flux is hopefully conserved even in cutting off the jet tip. The execution of cut-off operation is judged by

$$\gamma \leq \beta_0 \quad \text{at} \quad P_2 \tag{17}$$

The shadow area in Fig.3 is removed when the above condition is violated. So the contact angle γ is always monitored during computations. The results by Dobrovol'skaya [1] are adopted as the threshold angle β_0 in the present work.

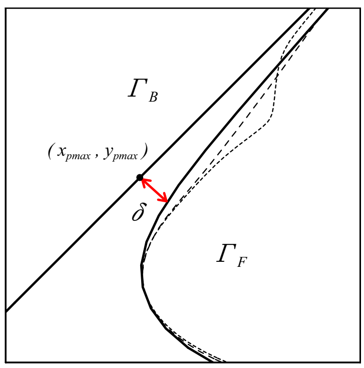

Fig. 3: Transitional phase due to gravity effects.

By the way, one of troubles in the computational simulation is the fact that the free surface shape with high curvature is made near the spray root, which is the stage before the flow exhibits the features of the self-similarity. The monotonous shape of the jet region tends to be deformed into small waved shapes. This is not a favorable tendency to examine the gravity effects in the jet flow simulation, because such shapes are easily evolved to the projectile of fluid into the air. Due to

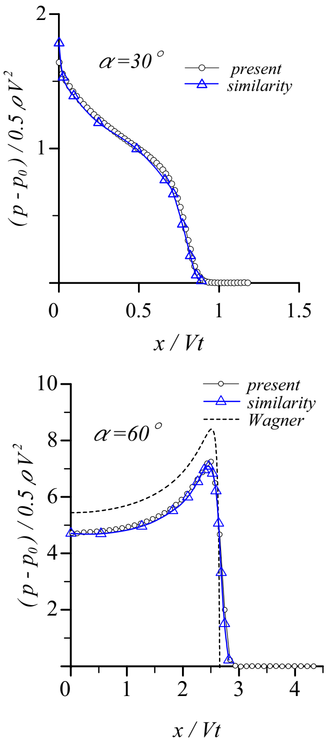

Fig. 4: Pressure distribution (without gravity)

the discrepancies due to singularities at the vertex are recognized in the case of $\alpha = 30°$. On the other hand, including the gravity term, we can simulate the evolution of the free surface from the jet formation up to the re-entry against the underlying free surface. Such computational results in case of $\alpha = 60°$ are shown in Fig.5.

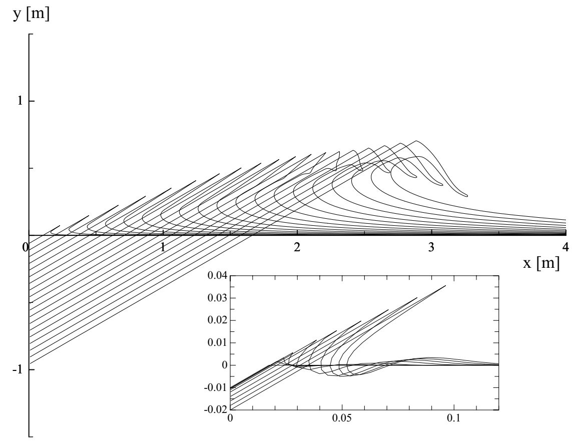

Fig. 5: Evolution of free surface due to water entry of a wedge with $\alpha = 60°$, $V = 1.0\,\mathrm{m/s}$ (Lower computational results correspond to initial 100 steps.)

the abrupt motion of a wedge into the undisturbed water surface, the concave water surface is formed near the spray root, which becomes more remarkable as the dead rise angle gets small, for instance, as shown in Fig.5. As the fluid particles need to flow along such a curve, these normal acceleration increases locally and results in a overshot curve. The pressure values become much higher on the body near that area temporarily. This moment corresponds with the occurrence of maximum peak pressure which is observed in the pressure time histories in the cases of small dead rise angles, although we can explain that it is air cushioning effects particularly for dead rise angles less than 10 degree. Therefore, this computational treatment is important for the prediction of the maximum peak pressure. Related with the convexity of the free surface, the following condition is adopted for this work.

$$\frac{\partial^2 x}{\partial s^2} \geq 0 \quad \text{for} \quad y \geq y_{pmax}\,, \;\; \delta \leq \delta_c \qquad (18)$$

where $y_{pmax} = Vt(\pi/2 - 1)$ by asymptotic theory, and δ means the thickness of the jet region and its critical value δ_c is set to $1.5\,l\tan\beta_0$ using jet region length l.

All computations are started from the state that a wedge is initially submerged into water slightly. The computed pressure distribution on the wedge surface are shown in Fig.4. The results by the present method give good agreements with similarity solutions in the case of neglecting the gravity term in equation (3), although

References

[1] Dobrovol'skaya, Z. N. 1969 On some problems of similarity flow of flow with a free surface. *Jonal of Fluid Mechanics*, Vol.36, pp.805-829.

[2] Faltinsen, O. M. 2002 Water entry of a wedge with finite deadrise angle. JOUNAL OF SHIP RESEARCH, Vol.46, No.1, March, pp.39-51.

[3] Fontaine, A. and Cointe, R. 1997 : A slender approach to nonlinear bow waves, *Philosophical Trans. of the Royal Soc.*, Series A, Vol.335, pp.565-574.

[4] Greenhow, M. 1987 : Wedge entry into initial calm water, *Applied Ocean Research.*, Vol.9, No.4, pp.214-223.

[5] Iafrati, A. and Korobkin, A. A. 2002 Hydrodynamic loads at the early stage of a floating wedge impact. *Proceedings*: 17th Intl. Workshop Water Waves and Floating Bodies.

[6] Iafrati, A. and Battistin, D. 2003 Accurate description of the jet flow developing during water impact. *Proceedings*: 18th Intl. Workshop Water Waves and Floating Bodies.

[7] ISOPE 1999 Report of the 2nd workshop of ISOPE Numerical Wave Tank Group. http://www.isope.org/conferences/conferences.htm

[8] Yim, B. 1985 Numerical solution for two-dimensional wedge slamming with a nonlinear free-surface condition, *Proceedings*: 4th International Conference on Numerical Ship Hydrodynamics, pp.107-116.

[9] Zhao, R. and Faltinsen, O. 1993 Water entry of two-dimensional bodies. *Jonal of Fluid Mechanics*, Vol.246, pp.593-612.