A NUMERICAL STUDY FOR BOW FLARE SLAMMING ANALYSYS USING BOUNDARY ELEMENT MODEL

Hajime KIHARA

Department of Mechanical Systems Engineering, National Defense Academy, Yokosuka, JAPAN

ABSTRAC

In this study, computational models using the boundary element method are proposed for the bow flare slamming analysis. This enables us to simulate nonlinear free surface flow due to the impact and compute the impact pressure on a bow section. The capabilities are demonstrated though numerical studies on the water impact problem of actual bow sections.

KEY WORDS: Slamming, bow flare, water impact, BEM, jet flow, splash, Post-Panamax container ship, Model 5415

INTRODUCTION

Wave loads for a running ship in waves is still a challenging problem in the research field of the naval architecture. In particular, the prediction of wave impact loads due to the slamming is important for the strength design and the safety of a ship. However, for the rational prediction of the ship behavior in severe sea conditions, the integrated evaluation of nonlinear wave load is indispensable. For example, the wave impact loads, the wave-induced motions and the structural response should be discussed coincidently in the prediction. The elucidation of nonlinear wave-body interaction is of growing importance from such a viewpoint. Traditionally the water impact problem is applied to the slamming analysis. So far enormous works have been studied (e.g. Korobkin(1996), Faltinsen(2005)). This approach is classified roughly into two. One is an analytical approach with mathematical models. Although the Wagner theory is well known, the recent studies are devoted to deal with the slamming problem of the arbitrary shaped body (Korobkin(2005), Takagi et al.(2007)). The other is a numerical one represented by the boundary element method (BEM). This is one of useful tools for such nonlinear analysis. A certain level of success was already achieved previously (Lin et al.(1984), Yim(1985), Greenhow(1987)), but the situation is moving to a new phase where the numerical simulation of the impact jet is realized (Zhao(1996), Battistin et al.(2003), Iafrati et al.(2004), Kihara(2004, 2006), Sun et al.(2007)).

The jet flow and the splash ejection on the water impact are characteristic fluid phenomena, while these features make the numerical simulation of the free surface flow more complicated. Conventionally the robust computation in the problem was not always easy because the numerical description corresponding with such phenomena was required. In the present paper, some numerical studies on the water impact problem of two-dimensional sections are carried out using the BEM, which can resolve the influence of the actual body shape on the hydrodynamic pressure or that of complicated geometry such as a flare, a bulbous bow, a combination and so on.The purpose of the present study is a demonstration that the numerical procedure using the BEM is available to the slamming analysis of a ship, though obtained results are within the framework of the potential theory. Fully nonlinear computation in time domain is employed, but the present approach doesn't aim at the simulation of the flow separation, even though it is the non-viscous one. Generally it is known that the pressure on the body surface shows a small negative value in the jet region. Although it is an idea to judge the flow separation by its sign, we don't take this choice in the present study. This is because the change of sign is too small and sensitive to judge. Therefore the computation is performed without considering any computational treatment of fluid reattachment with the hull surface after the flow separation. The application of the developed computational models would enable the prediction of the wave loads acting on the arbitrary sections of a ship.

FORMULATION OF THE PROBLEM

As shown in Fig.1, we consider the problem of a symmetrical body penetrating the water surface. The y and z axes are taken along the undisturbed free surface and along the body's centerline pointing upward, respectively. Only one half of the fluid domain for $y > 0$ is studied due to the symmetrical property of the problem about the z axis. It implies that the problem corresponds to the vertical motion of a hull section in heading or following waves, in addition, the vertical speed in the figure can be considered the body's relative velocity with the water surface of incoming waves at the target section. The fluid domain is surrounded by the boundaries consisting of the free surface, the body surface, the centerline, the side walls, and the bottom.

Assuming the fluid is incompressible and the flow is irrotational, the fluid motion is specified by the velocity potential. The problem of the velocity field is governed by the following equations:

$$\frac{\partial^2 \phi}{\partial y^2} + \frac{\partial^2 \phi}{\partial z^2} = 0 \quad \text{in} \quad \Omega . \tag{1}$$

Surface tension is neglected and the kinematic and dynamic boundary conditions of the free surface are

$$\frac{D\boldsymbol{x}}{Dt} = \nabla \phi \qquad \text{on} \quad \Gamma_F , \tag{2}$$

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

where $D\,/Dt$ means the substantial derivative and g is the acceleration of gravity. A position vector in the domain is denoted by $\boldsymbol{x} = (\,y, z\,)$. The boundary condition on the body surface is

$$\frac{\partial \phi}{\partial n} = \boldsymbol{v}_B \cdot \boldsymbol{n} = -Vn_z \qquad \text{on} \quad \Gamma_B \tag{4}$$

where $\boldsymbol{v}_B = (\text{-V}, 0\,)$ is the velocity of a body motion and $\boldsymbol{n}$ is the normal vector on the body pointing into the fluid. The boundary conditions on the other boundaries are

$$\frac{\partial \phi}{\partial n} = 0 \qquad \text{on} \quad \Gamma_W, \Gamma_C, \Gamma_0 . \tag{5}$$

The hydrodynamic pressure can be computed by using the Bernoulli equation. On that occasion, the time derivative of the velocity potential is necessary. Although it can be obtained by using the finite difference method to the velocity potential, the computational accuracy of the pressure is generally said to be not good. For more accurate prediction, the boundary value problem as for $\phi_t = \partial\phi/\partial t$ is also considered in the present study. The governing equation is also Laplace equation as follows:

$$\frac{\partial^2 \phi_t}{\partial y^2} + \frac{\partial^2 \phi_t}{\partial z^2} = 0 \quad \text{in} \quad \Omega . \tag{6}$$

The free surface condition for ϕ_t can be written as

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

The condition on the body surface for ϕ_t is given by

$$\frac{\partial \phi_t}{\partial n} = \kappa |\nabla \phi|^2 + \frac{\partial \phi}{\partial n}\frac{\partial^2 \phi}{\partial s^2} - \frac{\partial \phi}{\partial s}\frac{\partial}{\partial s}\frac{\partial \phi}{\partial n} \quad \text{on} \ \Gamma_B , \tag{8}$$

where $\partial\,/\partial s$ is the tangential derivative along the the boundary, and the κ denotes the local curvature of the body contour. The boundary conditions on the other boundaries are given by

$$\frac{\partial \phi_t}{\partial n} = 0 \qquad \text{on} \quad \Gamma_W, \Gamma_C, \Gamma_0 . \tag{9}$$

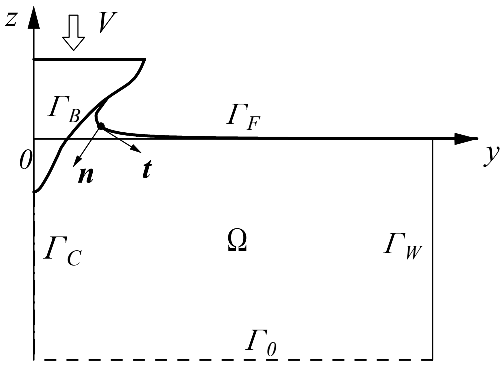


Fig.1 Definition of coordinate system

The boundary value problems are formulated as above. The initial conditions of the free surface would complete the problems. Then we can formulate the water impact problem as the initial-boundary value problem. On these matters we discuss in later section.

BOUNDARY ELEMENT MODEL

Two sets of boundary value problems in equations (1)-(5), (6)-(9) are solved using the BEM. Our adopted procedures in solving the boundary value problem are summarized as follows:

- Using linear isoparametric elements for discretization
- Arranging double nodes on corner points
- Evaluating the element integration analytically
- Excluding the bottom boundary by using a mirror image
- Imposing rigid wall condition on the side wall far away from a body
- Making the nodal density locally higher near the body

The water impact of a body causes the jet flow which tends to form the thin fluid layer on the body surface in the numerical simulation. Therefore we have to deal with the fluid domain with such a particular shape in the BEM computation for the water impact problem. As for the computation of shape influence coefficients, we need to keep the computational accuracy especially in the case of the particular geometric shape, as mentioned by Kihara (2004). This computational accuracy makes or breaks to the robust computation.

Starting from the initial condition, the computation can be made proceed by updating the moving boundary every time-step. So the mixed Eulerian-Lagrangian method is employed in the present study. The adopted procedures in the computation of moving boundary are summarized as follows:

- Using the 4th order Runge-Kutta scheme for the time integration of the free surface conditions
- Rearranging the nodes under the quasi-uniformity condition every several time-steps

NUMERICAL TREATMENT

Truncation Technique by Domain Decomposition

The impulsive motion of a body induces the fluid behavior with large acceleration and then the fluid runs up along the body surface very quickly. Theoretically the velocity is singular at the intersection in a moment of the water impact. It means that the velocity is infinite there. However, the computed velocity values based on the nonlinear theory are finite, and numerical errors are inherent in the computation for short duration after the water impact. It implies that solutions are not unique. Considering the actual fluid phenomena, the fluid is disintegrated into liquid fragmentation, which is no longer the continuous fluid domain. We can trace the motion of a jet tip to some extent, but computational efforts will be added more and more because of extending a computational domain. Additionally such a situation leads to the numerical instability and the small negative pressure, and finally the computation terminates with the overlap of the boundary. Moreover, the continuous fluid ejection by the splash or spray, which breaks out even in the case without the slamming, also results in the computational termination.

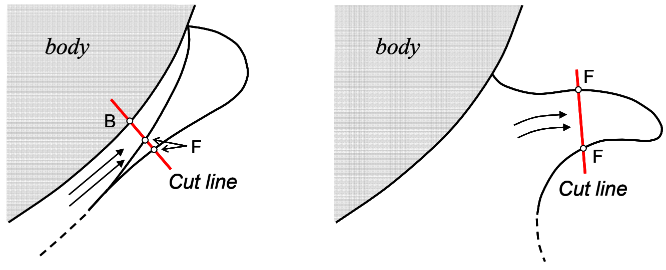
From the above-mentioned considerations, we introduced the idea of artificial liquid breakup to the scheme, that is to say, the truncation technique of the jet flow and splash ejection (Kihara(2006)). This technique is very effective to the simulation of the free surface deformation in slamming and also applied to the present study in an improved manner.

Our concept of the truncation technique is shown in Fig.2, where symbols F and B denote a point on the free surface and one on the body surface, respectively. Although two kinds of free surface shapes are described in (a) of the figure, both may not occur in the same scale actually. When the fluid is truncated, the velocity potential on the interface can be computed by using the idea of the domain decomposition (D.D.) method. In this approach, there are alternative ways, one is the rational D.D. method using the BEM and the other is the practical one which is based on the shallow water approximation, as shown in Fig.3.

For executing the domain decomposition exactly in the former manner, the adequate boundary condition is necessary to be imposed on the interface boundary. As shown in Fig.4, introducing the double nodes on the corner, which corresponds to a point F or B in Fig.3, we can make both domains matched on the interface boundary accurately. Generally there is concern about less computational accuracy in using the domain decomposition method by the BEM, than the computational accuracy in using the BEM in the single domain. However, this is mainly because the matching conditions cannot be fully satisfied for the flux value on the corner. It occurs particularly in using the constant elements. Notably the use of linear elements also brings such a merit. Matching conditions at a junction are described as follows:

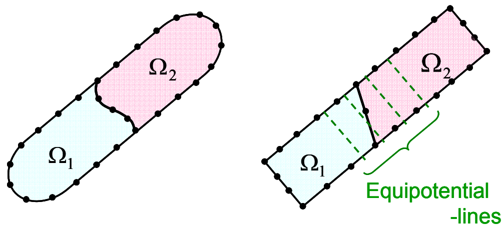
(i) On the boundary with Neumann condition

$$(\phi)_{j+1} = (\phi)_j \qquad \text{at} \quad \text{node } j+1$$
$$(\phi)_k = (\phi)_{k+1} \qquad \text{at} \quad \text{node } k \qquad (10)$$



(a) Truncation between F and B (b) Truncation between F and F

Fig.2 Conception diagram of truncation technique



(a) Rational D.D. using BEM (b) Practical D.D. based on shallow water approximation

Fig.3 Conception diagram of Domain-Decomposition

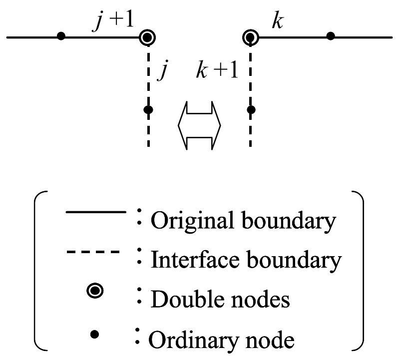


Fig.4 Node arrangement for matching domain

(ii) On the boundary with Dirichlet condition

$$(\phi)_{j+1} = (\phi)_j \qquad \text{at} \quad \text{node } j+1$$
$$\left(\frac{\partial \phi}{\partial n}\right)_k = \left(\frac{\partial \phi}{\partial n}\right)_{k+1} \qquad \text{at} \quad \text{node } k \qquad (11)$$

(iii) On the interface boundary

$$(\phi)_j = (\phi)_{k+1} \qquad \text{at} \quad \text{node } j$$
$$\left(\frac{\partial \phi}{\partial n}\right)_j = \left(\frac{\partial \phi}{\partial n}\right)_{k+1} \qquad \text{at} \quad \text{node } k+1 \qquad (12)$$

where the index symbols such as j and k correspond to the node number in Fig.4. Although the condition that two potential values are equal is basically imposed on the double nodes, we might have to exchange the potential

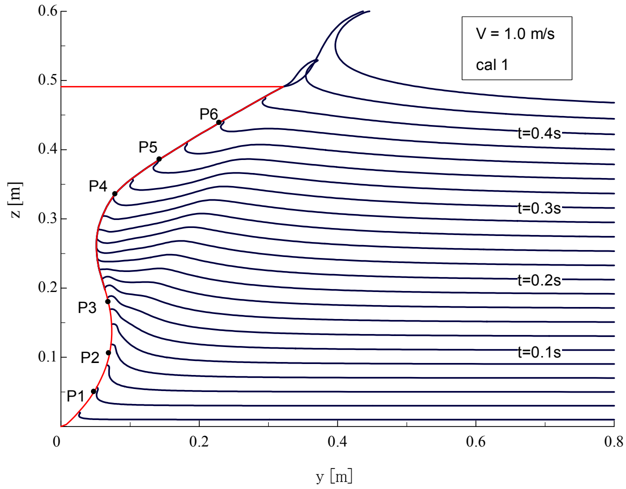


Fig.5 Computed free surface evolution in water entry

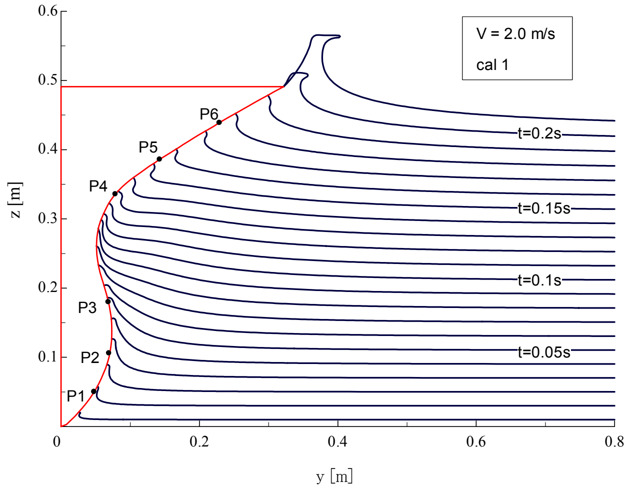


Fig.6 Computed free surface evolution in water entry

condition for another condition that two flux values are equal. This is a key point particularly in matching on the Dirichlet boundary.

On the other hand, the latter is the practical manner by the interpolation, where all potential values on the interface for splitting the domain into two are interpolated using computed results. This is a more simple manner and brings satisfactory results practically, which was called the cut-off model and already made sure of by Kihara(2004). In principle, it works reasonably in the simple flow like uniform flow. The domain decomposition by the coupling computation for two domains is introduced only at the first time step, while the practical approach based on the interpolation is adopted in computing afterward in the present study.

Initial Condition in Water Impact Problem

The initial condition of the free surface is necessary to complete the problem. Assuming that the small portion of the body is already submerged into water and there is no initial disturbance on the free surface, such condition is given by $\phi = \phi_t = \zeta = 0$ on Γ_F, where ζ denotes the free surface elevation. However, these conditions are not always adequate as the impact phenomena break out in a moment of body motion. The treatment of the initial condition is important to discuss the nonlinear wave-body interaction due to the impulsive motion in which impulsive acceleration is induced, because the prediction of the hydrodynamic pressure on a body is affected at the early stage of the impact. So the initial condition by an analytical manner is studied by Iafrati and Korobkin(2004). In the present study, we propose the application of self-similarity solution of a wedge body as the initial condition of an arbitrary shaped body. We decide the dead rise angle corresponding with the inclination of a target body surface at water line. Although this approach is theoretically inconsistent, it is used only at the first step and the nonlinear solutions satisfying exact boundary conditions are found out in afterward continuing time steps. According to our experience, we can say this approach is practical. In introducing this approach, we can use the idea of the already-mentioned truncation technique to joint the free surface of the self-similarity solution with the body surface.

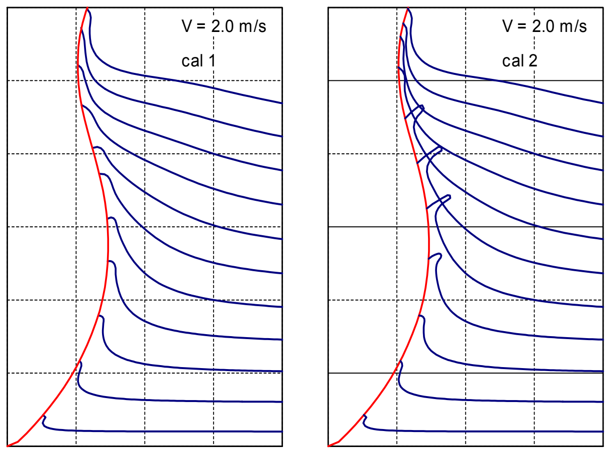


Fig.7 Comparison of computational approaches in dealing with flow detachment

NUMERICAL EXAMPLES

As feasibility studies, the present computational model is applied to the water impact of the bow section. Two kinds of ships are used. One is the ship whose bulbous bow is not so large, but the flare angle is large, and the other is the ship whose flare angle is small, but there is a sonar dome under the bow. Both have the complex geometric shape where convex and concave parts are combined. As far as we know, there are few examples that the BEM computation has been applied to such a shape.

In advance of the discussion of results, we mention to the computational condition briefly. On the size of computational domain, horizontal width from the center boundary Γ_C is set to 10.0 m, and the water depth is set to 4.0 m. Total number of nodes is initially about 350 points on the whole boundary, of which about 240 points are used on the free surface and 40 points are used on the body surface. The nodes on the body are increased as the wetted surface extends, and finally 80 points are used. Time step size is 5×10^{-5} sec in the case of V=2.0 m/sec. All information is nearly common in both numerical examples.

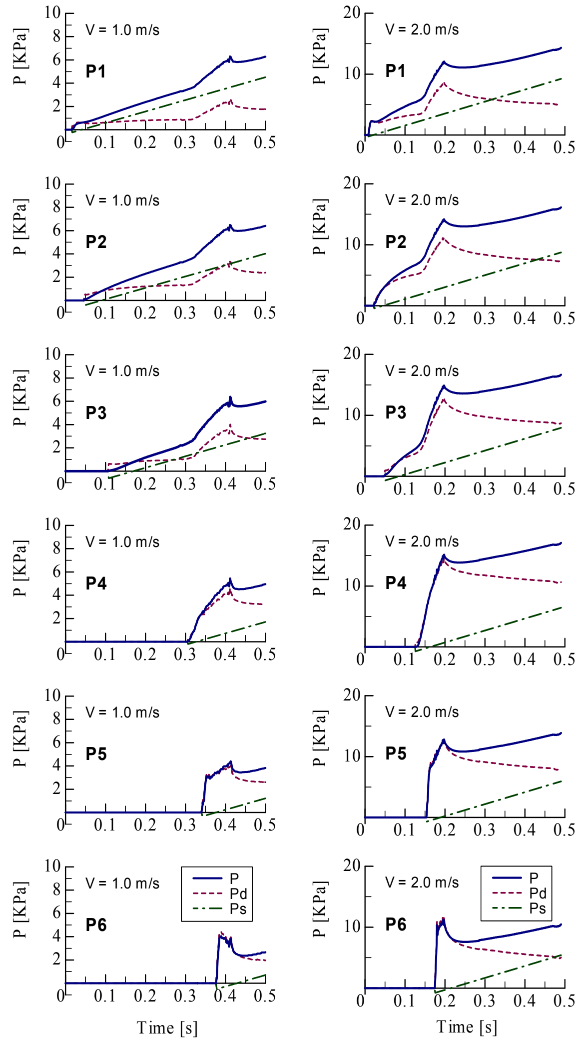


Fig.8 Computed pressure time histories

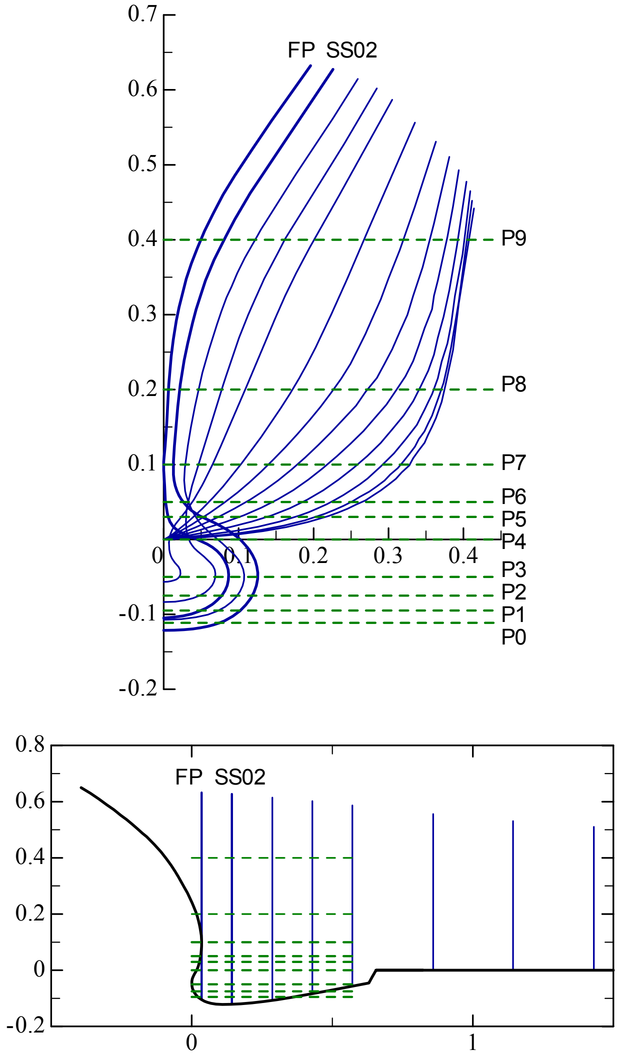


Fig.9 Body plan and profile of the Model 5415

Bow Section of a Post-Panamax Container Ship

With the recent enlargement of ships, the container ship has a large flare for holding broader deck area. Since it is subject to large impact load on the flare in heavy sea condition, it is important to estimate bow flare impact pressure correctly. This problem is recently studied by Ogawa et al.(2006), who estimate the impact pressure as the form of the time variation of added mass in the nonlinear strip method and show the good agreement with measured pressure.

For the same shape as the bow section of the container ship, the simulation of water entry is implemented. Computational results of the free surface evolution are shown as Fig.5 and Fig.6. Two-thirds of computational nodes on the free surface are used for the description of the free surface profile with the same length as the wetted surface, so satisfactory results in capturing the free surface deformation due to the generation of the jet flow are confirmed. As the truncation technique is introduced, there is no numerical description like the development of thin fluid layer on the body surface, the splash or spray ejection and the fragmentation of fluid. In the case of Fig.5 (V=1.0 m/s), we can recognize weak wave propagation by the body motion, so the apparent dead rise angle becomes smaller than that in calm water due to the wave slope. On the other hand, the impact phenomenon is more remarkable than the wave propagative one in the case of Fig.6 (V=2.0m/s). The fluid flows at once into the concave part between the flare and the bulb, so the apparent dead rise angle is larger.

Introducing different treatments of the truncation technique, we can obtain the different results in the free surface deformation even to the same target, as shown in Fig.7. It suggests that the flow separation from the body surface should be caused even in non-viscous flow. However we recognize that the difference between both results on the hydrodynamic pressure is negligibly small.

The pressure is computed at each point shown in Fig.5 and Fig.6. These time histories are shown in Fig.8. When the flare's top penetrates water surface, it is confirmed that the maximal peak value is indicated at all points. Taking notice to the pressure on the flare, that is to say, on P4, P5 and P6, we can recognize that the increase of hydrodynamic component is much larger at P4 than the other two points, which seems to be the result to pay attention to. On the other hand, the abrupt increase which is characteristic in the

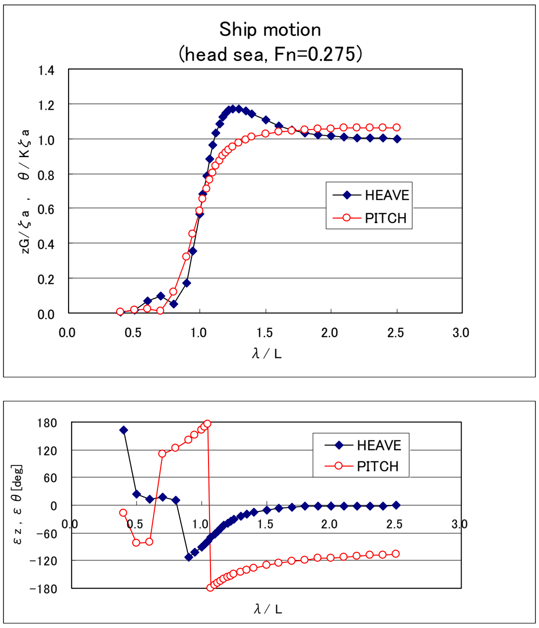


Fig.10 Computed ship motion of Model 5415

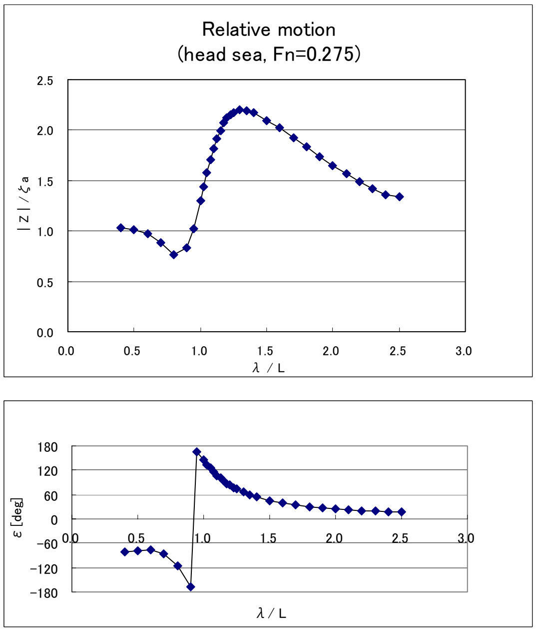


Fig.11 Computed relative motion of Model 5415

water impact can be recognized at P5 and P6. Finally, the computed pressure time histories have similar tendencies each other in two cases as for motion velocity.

Bow Sections of a Model 5415

From an academic viewpoint, the bow slamming of a battle ship is one of interesting problems, because it has a large sonar dome under the bow. In the present study, the Model 5415 is adopted for numerical analysis. The body plan and a part of the profile are shown as Fig.9. For slamming analysis, we need to know the critical condition that the slamming is induced in waves. In order to decide the motion velocity of a body, we compute ship motion and relative motion, as show in Fig.10 and Fig.11. Conventional strip method is used for the computation, but the computed result is not exact because the sonar dome is excluded from the computation. Additionally the dynamic and static swell-up is neglected. So the relative motion and velocity of the hull section with distance x away from the center of gravity are written by

$$\begin{cases} Z(t,x) = z_G(t) - x\theta(t) - \zeta_w(t,x) \\ \qquad \equiv \left|Z(x)\right| e^{i(\omega_e t + \varepsilon)} \\ V(t,x) = \dot{z}_G(t) - x\dot{\theta}(t) + U\theta(t) \\ \qquad \equiv \left|V(x)\right| e^{i(\omega_e t + \varepsilon_V)} \end{cases} . \tag{13}$$

The slamming occurrence can be judged with the following conditions:

$$\begin{cases} \left|Z(x)\right| > d(x) \\ \left|V(x)\right| > 0.09\sqrt{gL} \end{cases} , \tag{14}$$

where $d(x)$ is the draft at the target section. From the above procedures we can estimate the maximal relative velocity is about 2.0m/s in a model scale. In a real scale with ship length 155m, this corresponds to the running at Froude number 0.275 in heading waves of wave period 11.4sec with wave height 11.3m. Regardless of very rough estimation, the computational condition is set for the feasibility study.

Computational results of the free surface evolution are shown as Fig.12 and Fig.13. Although these are results if the bow section penetrates the water surface with constant speed, the computational model is functional even to the case of a section with a sonar dome. It is not the fact that splash and plunging breaker can be simulated but the fact that the computation is executable to such an extent, to be emphasized here. Finally the vertical force acting on the section is computed to two bow sections, is shown in Fig.14 and Fig.15. First we can recognize that the vertical force indicates the large peak in a moment of impact with section's bottom and its value becomes a maximum. However, the influence of flare on vertical force, especially on hydrodynamic component, tends to increase more and more as the motion velocity becomes large, though the flare angle is not so large in this example. This implies a possibility that the hydrodynamic component of wave loads may become comparable to the impact load. For the further discussion, more detailed investigation including experimental data is needed.

CONCLUSIONS

The computational models with the Boundary Element Method are proposed. The key techniques are also described for the practical numerical simulation. Moreover the capabilities of the approach are demonstrated though the numerical studies on the water impact problem for two kinds of ships. Computed results imply a possibility that the hydrodynamic component of wave loads may become comparable to the impact load. Future work is planned for the validity of the present approach and the strategy of slamming analysis by the approach.

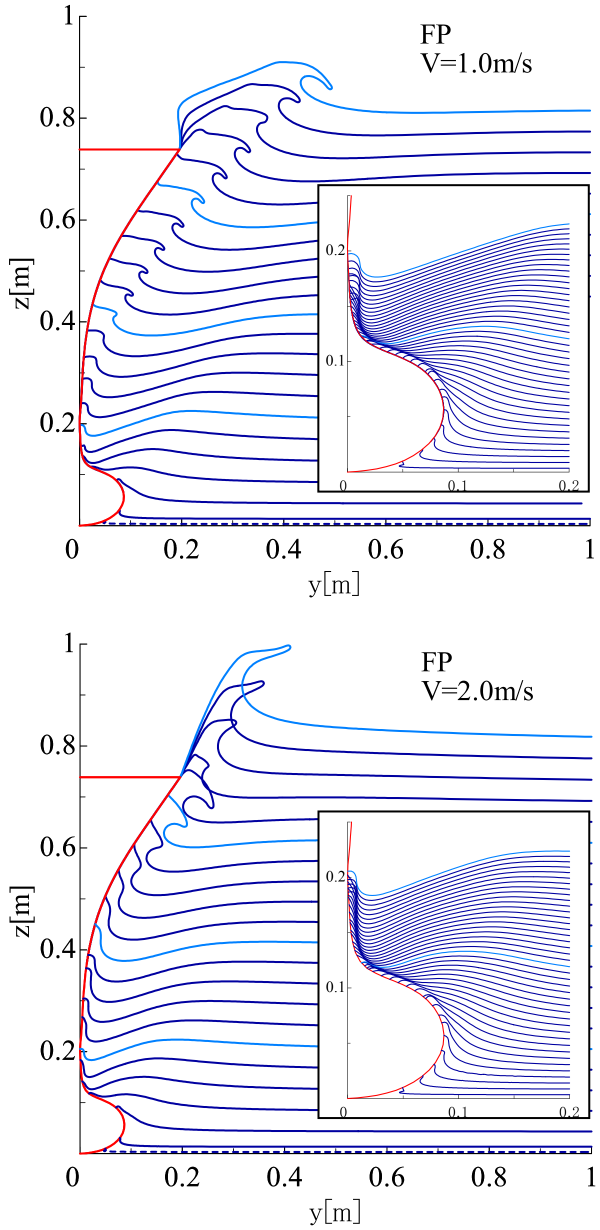


Fig.12 Computed free surface evolution in water entry

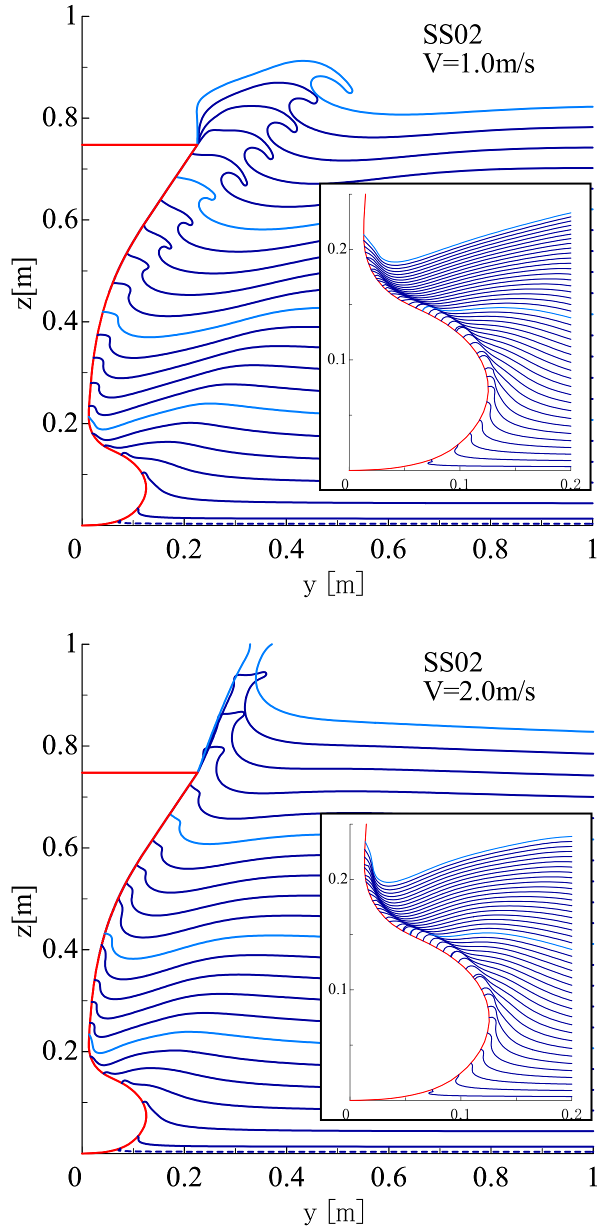


Fig.13 Computed free surface evolution in water entry

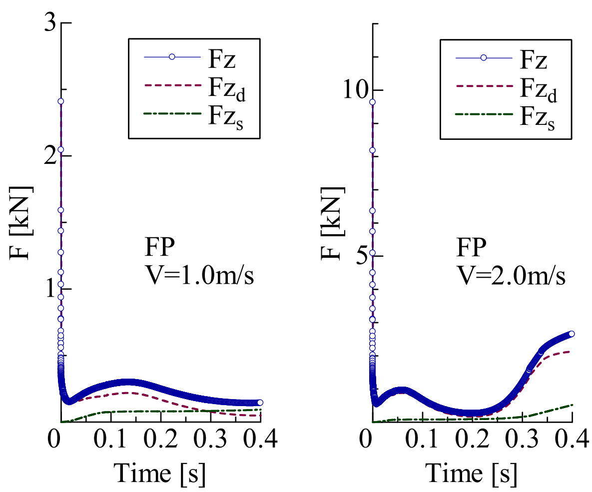


Fig.14 Vertical force on a bow section FP

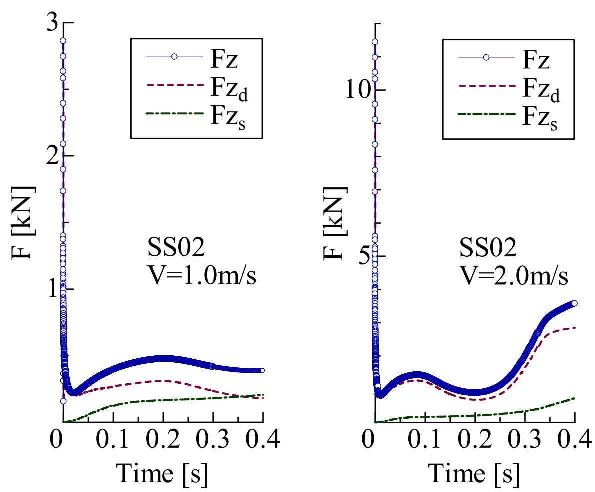


Fig.15 Vertical force on a bow section SS02

REFERENCES

Battistin, D. and Iafrati, A. (2003), “Hydrodynamic loads during water entry of two-dimensional and axisymmetric bodies”, *Journal of Fluid and Structures*, Vol.17, pp.643-664

Faltinsen, O.M. (2005), “Hydrodynamics of High-Speed Marine Vehicles”, *Cambridge University Press*

Greenhow, M. (1987), “Wedge Entry into initially Calm Water”, *Applied Ocean Research*, Vol.9, pp.214-223

Iafrati, A. and Korobkin, A.A. (2004), “Initial stage of flat plate impact onto liquid surface”, *Physics of Fluids*, Vol.16, pp.2214-2227

Kihara, H. (2004), “Numerical Models of Water Impact”, *Proc. of 4th International Conference on High -Performance Marine Vehicles*, pp.200－214

Kihara, H. (2006), “Computing Method for the Flow Analysis around a Prismatic Planing-Hull”, *Proc. of 5th International Conference on High -Performance Marine Vehicles*, pp.262－272

Korobkin, A.A. (1996), “Water impact problem in ship hydrodynamics”, *Advance in Marine Hydrodynamics, M. Ohkusu, Ed* (*Chap.7*), Computational Mechanics Publishing, Southampton, Boston

Korobkin, A.A. (2005), “Analytical Models of Water Impact”, *Euro. J. Applied Mathematics*, 16, pp.1－18

Lin, W.M.; Newman, J.N. and Yue, D.K. (1984), “Nonlinear Forced Motions of Floating Bodies”, *Proc. of 15th Symposium on Naval Hydrodynamics*, pp.33-49

Ogawa, M., Matsunami, R. and Arai, M. (2006), “Effect of Bow Flare Shape on Water Impact Pressures”, *Int. J. of Offshore and Polar Eng.*, Vol.16, No.2, pp.112-117

Sun, H. and Faltinsen, O.M. (2007), “Water Impact of Horizontal Circular Cylinder and Cylindrical Shells”, *Applied Ocean Research*, Vol.28, pp.299-311

Takagi, K. and Ogawa, Y. (2007), “Flow Models of the Flare Slamming”, *Proc. of International Conference on Violent Flows*, pp.173-179

Yim, B. (1985), “Numerical solution for two-dimensional wedge slamming with a nonlinear free-surface condition”, *Proc. of 4th International Conference on Numerical Ship Hydrodynamics*, pp.107-116

Zhao, R. , Faltinsen, O.M. and Aarsnes, J (1996), “Water entry of arbitrary two-dimensional sections with and without flow separation”, *Proc. of 21th Symposium on Naval Hydrodynamics*, pp. 408-423