Journal of Naval Architecture and Marine Engineering
December, 2014
http://dx.doi.org/10.3329/jname.v11i2.18724 *http://www.banglajol.info*

NUMERICAL SIMULATION OF WATER ENTRY OF DIFFERENT ARBITRARY BOW SECTIONS

P. Ghadimi[1*], M. A. Feizi Chekab[1], A. Dashtimanesh[2]
[1]Dept. of Marine Technology, Amirkabir University of Technology, Tehran, Iran, *Email: pghadimi@aut.ac.ir
[2]Faculty of Engineering, Persian Gulf University, Bushehr, Iran

Abstract:

Water impact phenomenon of general bow section is a critical event for planning hulls. In this paper, the water entry of several arbitrary bow sections is investigated. For this purpose, arbitrary bow shapes which are introduced by Lewis form approximation are considered. In order to obtain pressure distribution and free surface profile, volume of fluid (VOF) method coupled with finite volume method (FVM) are utilized in ANSYS-CFX solver. Pressure distribution, free surface, and evolution of intersection point on bow sections are presented, while secondary water impact is demonstrated. Comparison of the obtained results against previously published works shows good agreement.

Keywords: Water entry, arbitrary bow sections, pressure distribution, finite volume method, volume of fluid

1. Introduction

In earlier days, prediction of the hydrodynamic pressure acting on an impacting body was an important factor to study the slamming problem of a planing craft. It was actually realized that slamming on the bow section may cause structural damages (Yamamoto et al, 1985). The free surface flow occurring in transverse planes of high-speed planing craft has also strong similarities with the one generated during the water impact (Battistin and Iafrati, 2003). Therefore, an equivalent process is the water entry of a bow section. For these reasons, water impact analysis of 2-d sections is a very attractive problem. Accordingly, various solutions have been developed to analyze the water entry problem.

In 1929, von Karman (1929) introduced a significant work on this subject. He developed an analytical formula which allows the estimation of the maximum pressure on seaplane floats during water landing. Wagner (1932) modified the von Karman solution by taking into account the effect of water splash on the body. In the special case of wedges entering water vertically at a constant velocity, Dobrovolskaya (1969) derived a similar solution by making use of the simple geometry of the body.

Zhao & Faltinsen (1993) proposed a numerical model based on boundary element method (BEM) for the simulation of water entry of wedges. They removed the upper part of the generated jet flow by the so-called "cut-off" model, in which a new computational procedure is introduced at the jet root position. An interesting method that permits to keep the jet was also developed by Battistin and Iafrati (2004).

Water entry of circular section is also investigated by different authors. Greenhow (1988) studied the water entry of a rigid circular cylinder by using a boundary element method based on Cauchy's theorem. The water entry of a rigid circular cylinder is studied by Zhu et al. (2006) who used a CIP (Communicating Interacting Processes) method. Sun and Faltinsen (2006) have developed the two dimensional boundary element code to simulate the water flow and pressure distribution during the water impact of a horizontal circular cylinder. They satisfied the exact free surface boundary conditions.

The water entry of general sections has only been considered by few researchers. Here, the most important works that are available in the literature are reviewed. Finite Difference Method (FDM) was developed by Arai & Tasaki (1987) and applied by Arai & Matsunaga (1989) for the numerical solution of the water entry of a bow-flare ship section. The free surface was initially considered to be calm and the gravity effect was neglected. Later, the calculations for the water entry of different bow sections were conducted by Arai et al. (1995). It was found that the separated water flow can be generated by initial bottom slamming on a bow-flare section. This separated flow will impact on the bow flare at a later stage and very high forces may be acted on the bow. This is called the secondary water impact.

 Received on: April 2014

Drop tests of the bow sections have been performed by Aarsnes (1996). In the work done by Zhao et al (1996), one symmetric case which was considered by Aarsnes (1996) drop tests for the bow-flare section, has been numerically studied and good predictions were obtained. His model was utilized to study the flow about bodies having arbitrary shapes, and it was further developed to deal with the flow separation for bodies where the intersection point can be easily determined. Mei et al. (1999) also tried to use a generalized Wagner's method to study the drop tests by Aarsnes (1996), but the solutions can only be obtained for the region prior to where the flow separation from the knuckles appears. CFD methods such as FDM in Arai et al. (1995) and the CIP method used by Zhu et al (2006) seem to be able to perform this work, but up to now the simulations using most numerical methods have been too expensive to be used in a practical problem. More realistic solutions may be acquired by advanced numerical methods such as volume of fluid (VOF). Recently, some researchers have applied VOF to solve nonlinear free surface problems (Kleefsman et al., 2005), Kleefsman et al. (2005), Hargreaves et al. (2007), Ying et al. (2009), Wang and Wang (2010) and Rahaman et al. (2013).

Current authors have previously introduced analytical solutions for solving the water entry of wedges (Ghadimi et al., 2011) and circular sections (Ghadimi et al., 2012). Also, same authors have investigated the water entry problem using different numerical schemes, including VOF method for arbitrary bow sections (Ghadimi et al., 2013) and SPH method for symmetric and asymmetric water entry of wedges (Farsi and Ghadimi, 2014a,b) and water entry of catamarans (Farsi and Ghadimi, 2014c).

Water entry of different arbitrary bow sections can be considered as a main feature of the present article. The introduced general bow sections are based on the Lewis form approximation which was proposed Mei et al. (1999). Another aspect of the current work is the utilization of finite volume method in conjunction with volume of fluid in ANSYS-CFX, to solve the water impact problem of arbitrary bow sections. Furthermore, some physical aspect of the problem such as secondary impact is presented. For validation of the numerical solutions, the wedge water entry of 10 degree deadrise angle is considered and the results are compared with similarity solution presented by Zhao and Faltinsen (1993). Likewise, a general bow section that was one of test cases studied by Aarsnes (1996) and used by Zhao et al. (1996) and Mei et al. (1999) is also considered for validation purpose. In addition, the pressure distribution and the free surface profile due to the impact of the introduced arbitrary bow sections on the water surface are presented and analyzed. Evolution of intersection point is also studied.

2. Numerical Formulation

It is assumed that fluid behaves as a continuum rather than as discrete particles. Decomposing the Navier-Stokes equations into the RANS (Reynolds averaged Navier-Stokes) equations makes it possible to simulate practical engineering flows, such as water entry problem. For the numerical simulation of the problem, commercial code ANSYS-CFX is utilized. To discretize the governing equations, finite volume method (FVM) is used. Furthermore, to capture the interface between air and water, free surface, Volume of fluid (VOF) technique is applied.

The governing equations as well as numerical setup including initial and boundary conditions and some details are outlined in the following subsections.

2.1 Governing equations

The homogenous multiphase Eulerian fluid approach is utilized in ANSYS-CFX to describe the interface between the water and the air, mathematically. Both air and water share the same characteristic (in the free surface) such as velocity, turbulence, etc. The water and air must also be separated by a distinct resolvable interface. The governing equations that need to be solved by the ANSYS-CFX solver are the mass continuity and the momentum equation (Ahmed, 2011), which are given as

$$\frac{\partial}{\partial x_i}(\rho u_i) = 0 \tag{1}$$

$$\begin{aligned}\frac{\partial}{\partial x_i}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_i}(-\rho \overline{u}_i^{'} \overline{u}_j^{'}) \\ + \frac{\partial}{\partial x_j}\left[\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\delta_{ij}\frac{\partial u_l}{\partial x_l}\right)\right].\end{aligned} \tag{2}$$

In order to capture the sharp interface in hydrodynamic two phase flow problems, volume of fluid method is employed. Volume of fluid (VOF) technique, originally introduced by Hirt and Nichols, uses a color function named Volume Fraction (q). A transport equation (Eq. 3) is then solved for the advection of this scalar using the velocity field calculated from the solution of the Navier-Stokes equations at the last time step.

$$\frac{\partial q}{\partial t}+\vec{\nabla}.(q\vec{u})=0 \tag{3}$$

Numerical solution of Eq. 3 gives the volume fraction, *q*, for each phase (i.e. Air and Water) in all computational cells where $\sum_{k=1}^{2} q_k = 1$. In fact, distribution of the volume fraction (q) is as follows:

$$q=\begin{cases}1 & \text{for cells including fluid 1}\\ 0 & \text{for cells including fluid 2}\\ 0<q<1 & \text{for cells including the interface}\end{cases} \tag{4}$$

Using the volume fraction, an effective fluid with the variable physical properties is introduced as in

$$\begin{aligned}\rho_{eff} &= q\rho_1+(1-q)\rho_2\\ \upsilon_{eff} &= q\upsilon_1+(1-q)\upsilon_2\end{aligned} \tag{5}$$

where subscripts 1 and 2 represent two phases, i.e. water and air.

Furthermore, a *k-ε* turbulence model is utilized to consider the viscous effects. *k* is the turbulent kinetic energy and *ε* is the dissipation rate of the turbulent energy. The standard *k-ε* turbulent model in ANSYS CFX is as follows (Ahmed, 2011):

$$\rho u_j \frac{\partial k_i}{\partial x_j}=\frac{\partial}{\partial x_i}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right]+G_k+G_b-\rho\varepsilon \tag{6}$$

$$\rho u_j \frac{\partial \varepsilon_i}{\partial x_j}=\frac{\partial}{\partial x_i}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_i}\right]+C_{l\varepsilon}\frac{\varepsilon}{k}(G_k+C_{3\varepsilon}G_b)-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \tag{7}$$

where G_k and G_b are the generation of turbulent kinetic energy due to the mean velocity gradients and buoyancy. σ_ε, σ_k, $C_{l\varepsilon}$ and $C_{2\varepsilon}$ are the model constants and must be determined experimentally. μ_t and μ are also turbulent eddy viscosity and molecular dynamic viscosity, successively.

2.2 Initial and boundary conditions

When simulating the free surface flows, appropriate initial conditions and boundary conditions must be defined to set up appropriate velocity and volume fraction fields. It is necessary to create expressions using CEL (CFX Expression Language) to define these conditions. In the present article, the following conditions are set:

- The side walls are treated as planes of symmetry.
- Vertical velocities are prescribed at the lower inlet implying constant flux and forcing the water rise at the same velocity of the water entry.
- At the upper outlet, pressure is assumed to be zero.
- No-slip wall condition is imposed on the body surface.
- The initial free-surface level is set in a way that the interface touches the lowest point of the semi-cylinder at t=0.00 (s).
- A time step size of 0.0001s is used to catch all detailed information of solution process.

Furthermore, the size of the time step can be controlled using the stability and convergence criterion during the solution. The dimension of the computational domain is set as such that will significantly decrease the dependency of the solution on the boundary condition.

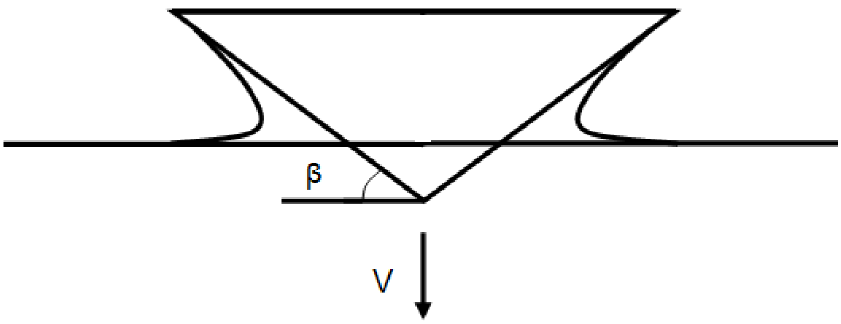


Fig.1: Water entry of a 10 degree wedge section

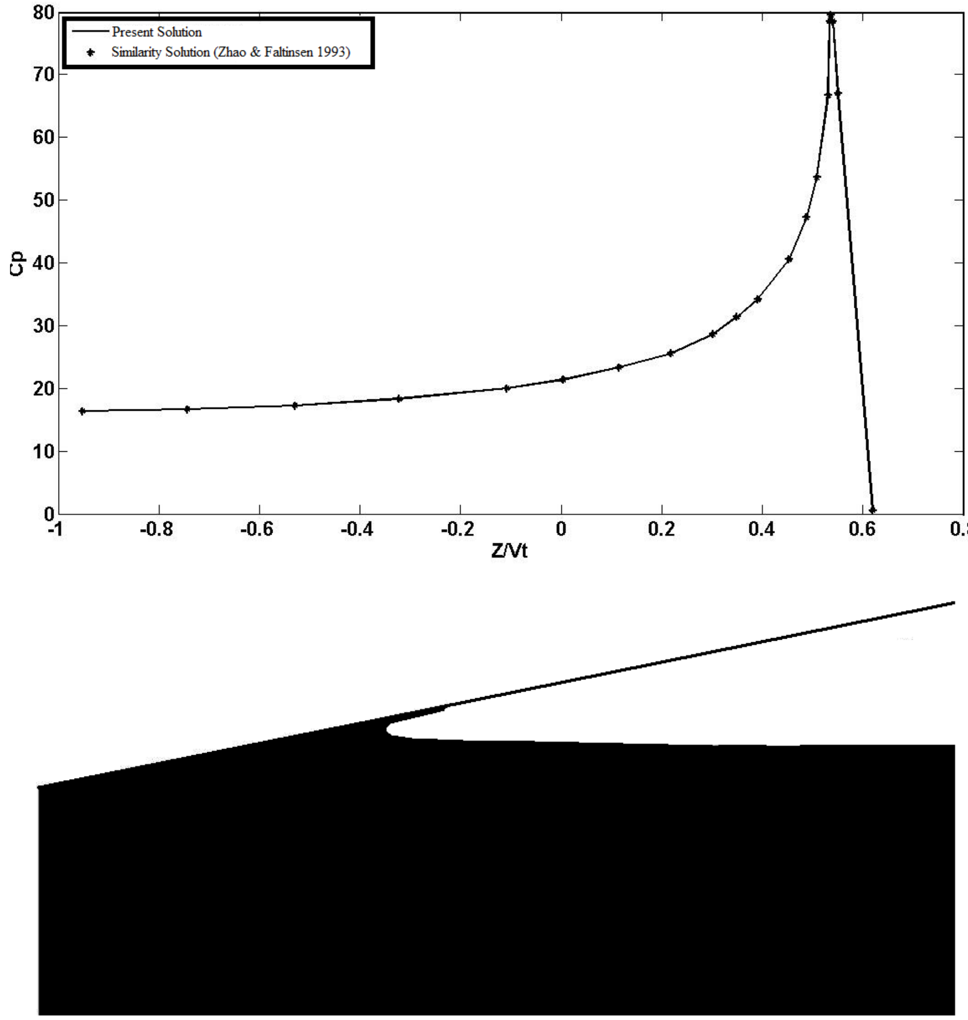


Fig. 2: Pressure distribution and free surface profile of a 10 degree wedge section

3. Validation

The coupled numerical solution of the water entry of arbitrary bow sections is verified in two different ways. At first, a 10 degree wedge section is entered into the water by a constant velocity. The resulting free surface profile and pressure distribution are compared with the existing results in the literature. Secondly, the arbitrary section which was one of test cases studied by Aarsnes (1996) is also investigated. The resulting pressure distribution and slamming force are compared with the numerical and experimental results of Zhao et al. (1996) and Aarsnes (1996) successively.

3.1 Water entry of a 10 degree wedge section

Analysis of the water entry of a 10 degree wedge section can be a basis to assure that the coupled numerical model can be implemented to investigate the water entry of arbitrary bow sections. Fig. 1 shows a wedge section which has 10 degree deadrise angle entering into the calm free surface. The water entry velocity is assumed to be constant during the impact.

The pressure distribution and free surface profile which are shown in Fig. 2 are compared against the results of Zhao and Faltinsen (1993) obtained by similarity solution. As observed in Fig. 2, the results are in good agreement with each other.

The water entry of an arbitrary bow section that was used by Aarsnes (1996) is also investigated to study the size of computational domain, time step and mesh quality that satisfy the numerical requirements for a successful simulation of the water entry of arbitrary sections.

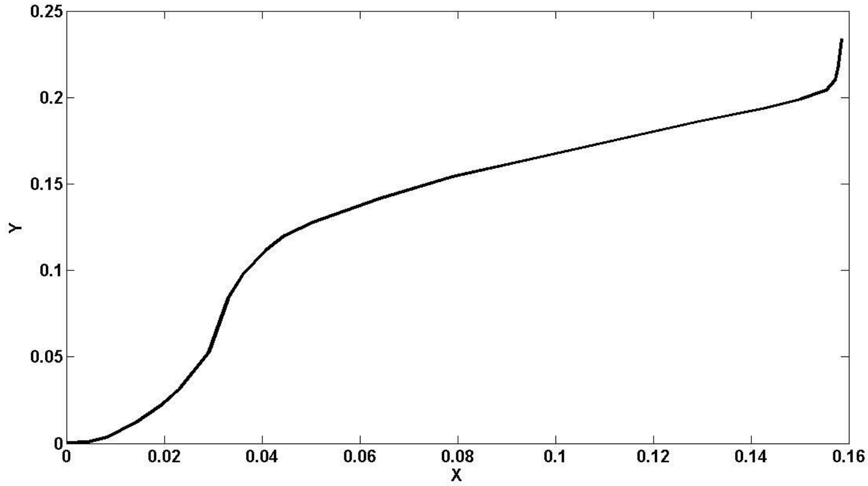


Fig. 3: The bow section considered by Aarsnes (1996) (consider the visual aspect ratio of the plot (y/x) is 2.5)

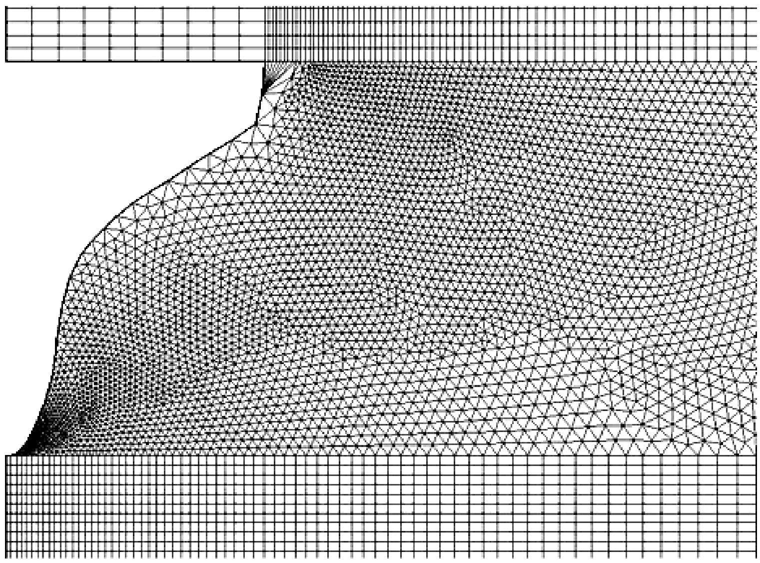


Fig. 4: Computational domain and the hybrid grid

3.2 Water entry of an arbitrary section

Fig. 3 shows a bow section which has been introduced by Aarsnes (1996). This profile is chosen in order to validate the numerical solver. In the numerical simulation, the section profile is formed by connecting the discrete points which are read from Fig. 3. A constant descending velocity is applied. The considered computational domain is a rectangular region whose length is 8 times larger than the maximum breadth of the body. A hybrid grid with finer cells near the body is also used leading to approximately 75320 cells in total as in Fig. 4. The reason for using hybrid grid is to better capture the horizontal free surface of water before the impact by using orthogonal grid and also to better model the curved shape of the hull by using a non-orthogonal triangular grid.

The computed pressure distribution and slamming force are compared against the numerical results obtained by Zhao et al. (1996) and experimental findings of Aarsnes (1996). It is clearly observed in Fig. 5 that our findings are in good agreement with the results of the fully nonlinear solution of Zhao et al. (1996). The free surface profile in which the maximum pressure takes place is displayed in Fig. 6. It can be observed that the maximum pressure occurs just before the spray roots of the jets reach the knuckles.

The resulting slamming force is also found to be in relatively good agreement with the experimental result of Aarsnes (1996) which is illustrated by the comparison shown in Fig.7.

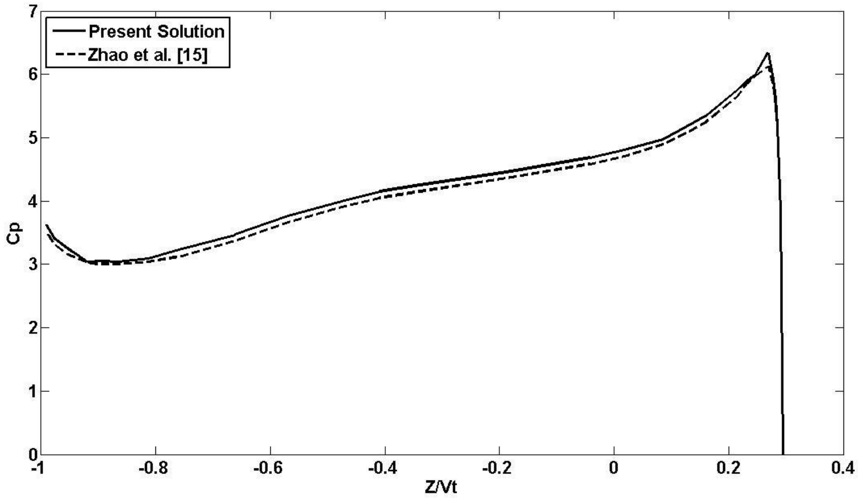


Fig. 5: Comparison of pressure distribution on the bow section

Fig. 6: Free surface profile just before the spray roots of the jets reach the knuckles

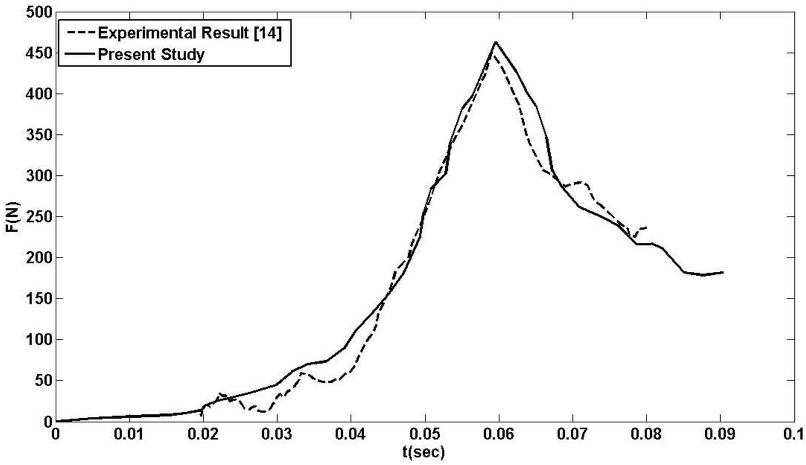


Fig. 7: Comparison of time history diagram of the slamming force

The numerical solver at this stage is applied on any arbitrary section. In the following sections, some new profiles are introduced and their corresponding pressure distributions as well as the free surface profiles are presented.

4. Introducing New Arbitrary Bow Sections

Some new bow sections are introduced in this section. Their design is based on the equivalent wedge sections and applying the Lewis form approximation (Mei et al, 1996).

4.1 Lewis form approximation

Lewis-form approximation can be used to map a section in the plane $w=p + iq$ to a closed form geometry in the plane $Z=y'+z'i$ (Fig. 8). The closed form geometry in the Z-plane can be considered as an approximation of several curves. For more details of this mapping, many reference books can be surveyed (Newman, 1977).

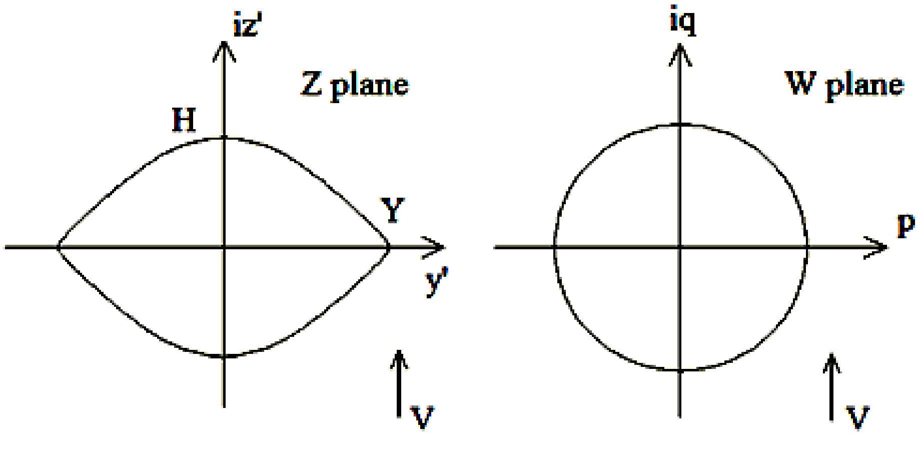


Fig. 8: Lewis form approximation

By implementing Lewis form definition, a rhombus with deadrise angle, α can be approximated as shown in Fig. 9. Now, if the particular section of the body which enters into the water surface, similar to that shown in Fig. 10 is considered, it is possible to define that section based on an equivalent wedge and using Lewis-form approximation, as demonstrated in Fig. 11. To find more details about Lewis-form approximation, readers are referred to Mei et al. (1999).

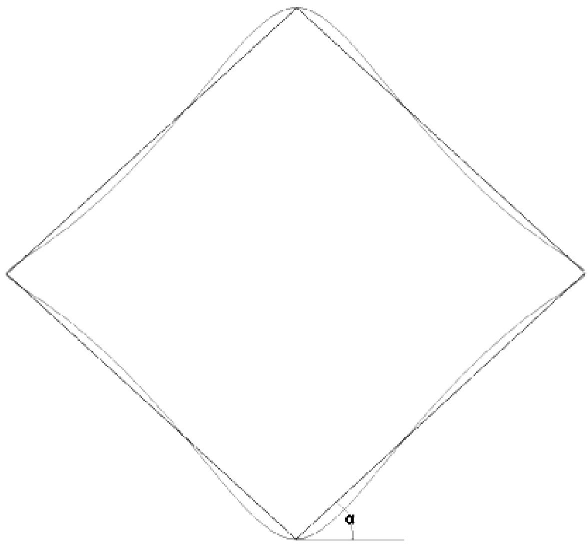


Fig. 9: A rhombus with deadrise angle, α approximated by using Lewis form

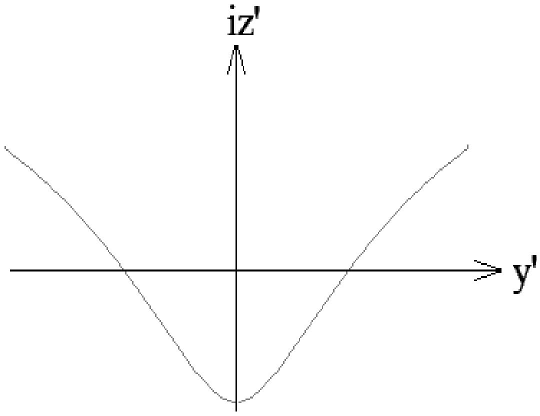


Fig. 10: The assumed entering body into the water surface

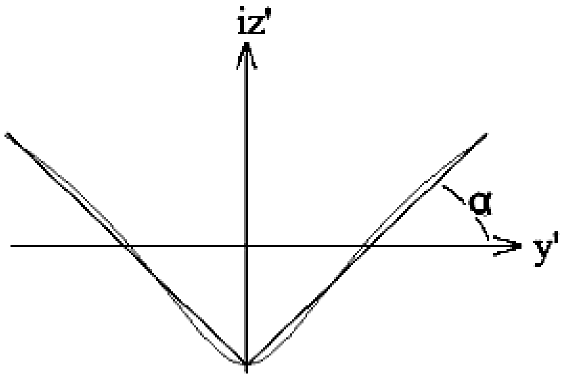
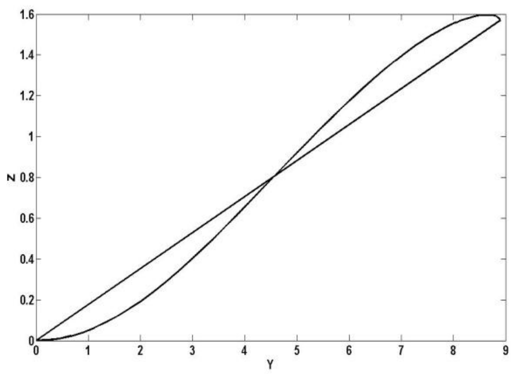


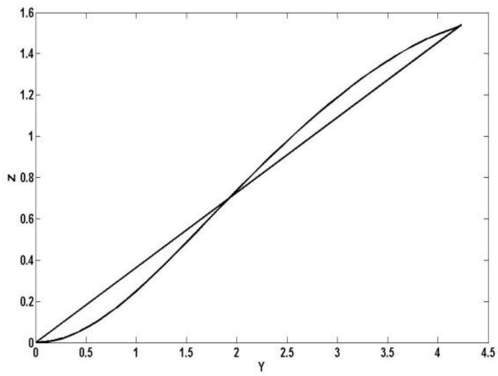
Fig. 11: Generation of a bow section based on an equivalent wedge section

5. Numerical Results

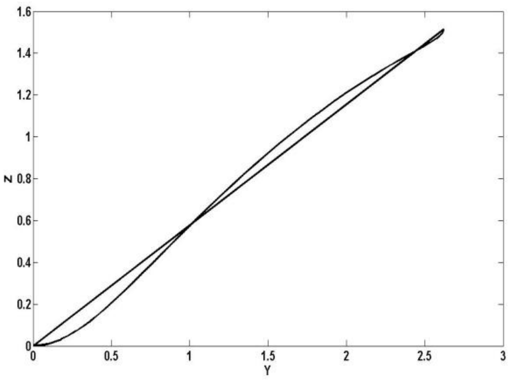
Based on the above explanation, five arbitrary sections are formed. These sections are equivalent to 10, 20, 30, 45 and 60 degree wedge sections as shown in Fig. 12.



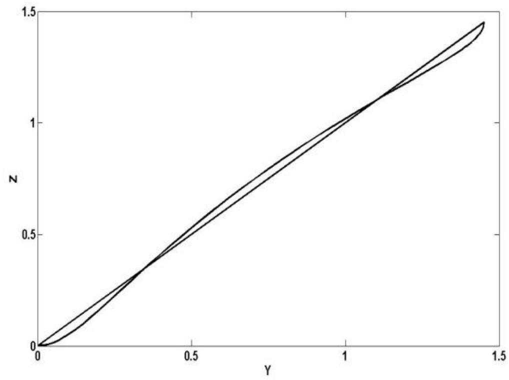
(a)



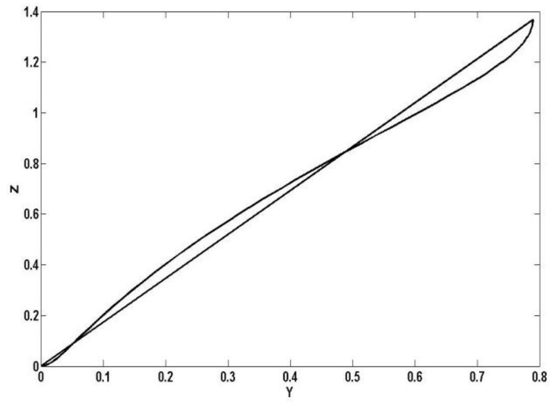
(b)



(c)

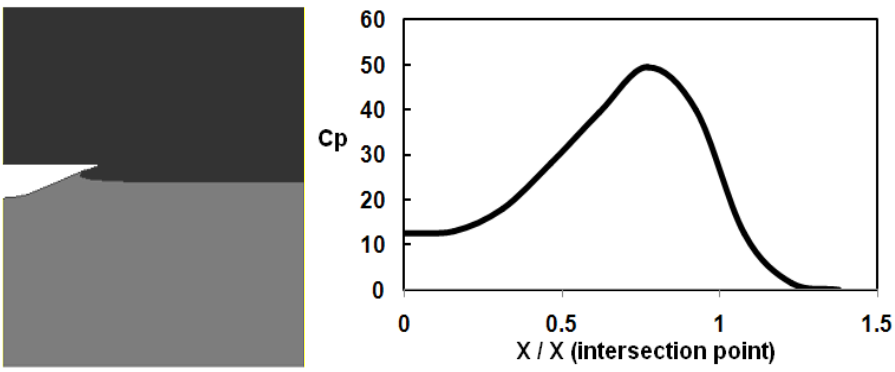


(d)

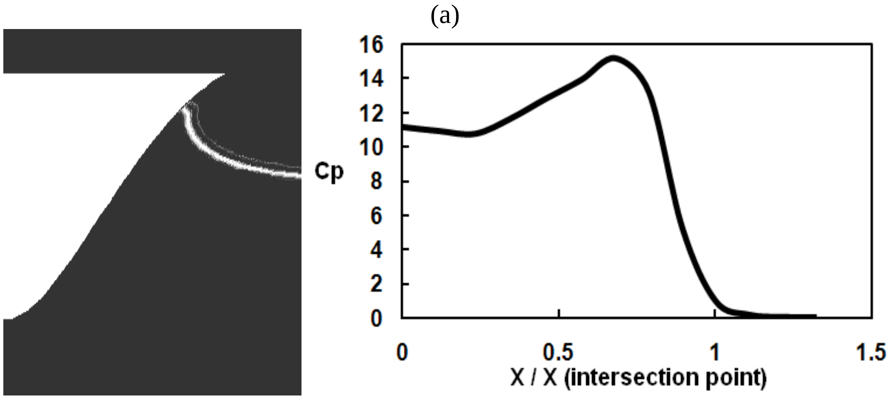


(e)

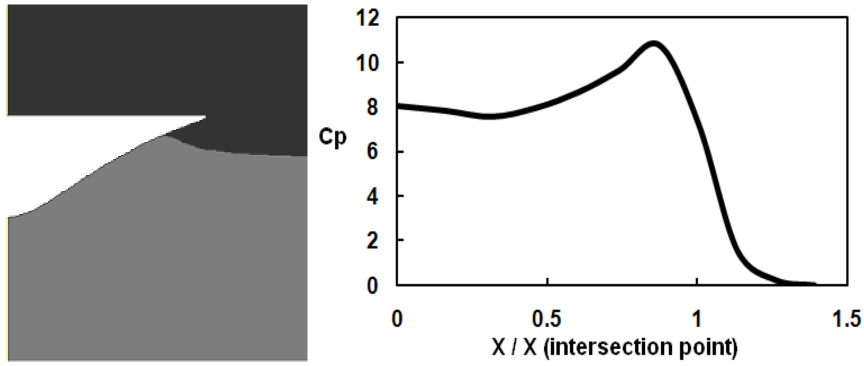
Fig. 12: The arbitrary sections based on different equivalent wedge section; (a) 10 degree, (b) 20 degree, (c) 30 degree, (d) 45 degree, (e) 60 degree.



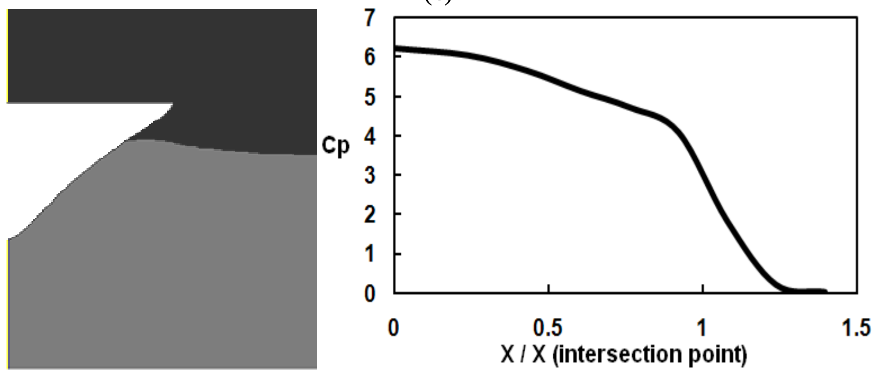
(a)



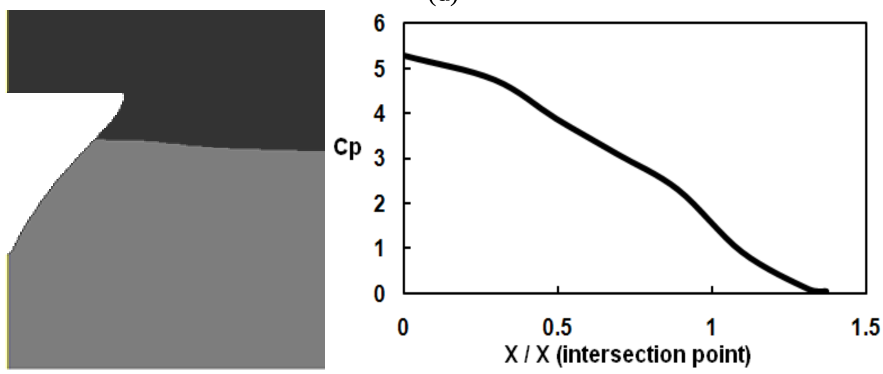
(b)



(c)



(d)



(e)

Fig. 13: The pressure distribution and free surface profile of the introduced bow sections; (a) 10 degree, (b) 20 degree, (c) 30 degree, (d) 45 degree, (e) 60 degree.

The arbitrary section which is equivalent to 60 degree wedge section has a minimum pressure distribution as observed in Fig. 13. By measuring the deadrise angle of the arbitrary section which is shown in Fig. 3, it is found that the deadrise angle of its equivalent wedge is approximately 55 degrees. The free surface profile related to each section is also depicted. Due to low impact velocity (0.58 m/s), the secondary water impact may not be generated. In reality, the secondary water impact is observed whenever the water flow separates from the body and once again impact on the body surface. However, for a particular case, the impact velocity of 2.43 m/s is imposed and consequently a secondary impact is observed. This secondary impact is illustrated in Fig. 14.

The issues pointed out above, are not the whole purpose of the current study. The present paper also tries to provide some understanding about the movement and characteristic of the flow near the intersection point. This information can contribute to the physical understanding of the problem. Therefore, Fig. 15 shows the position of the intersection point that is obtained by the coupled numerical model.

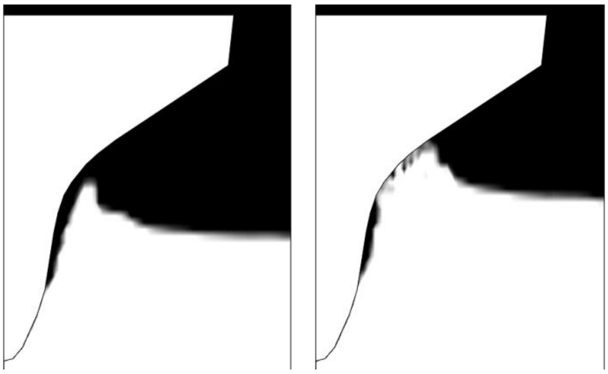
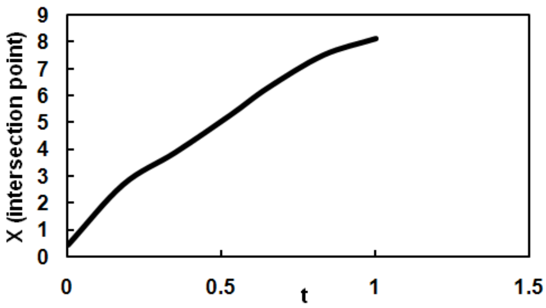
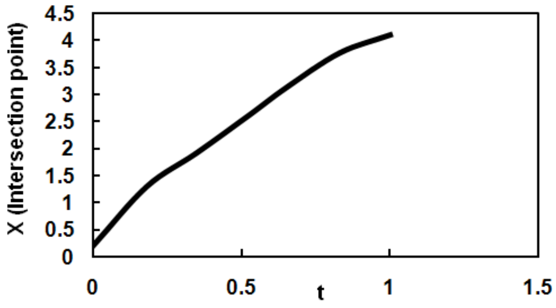


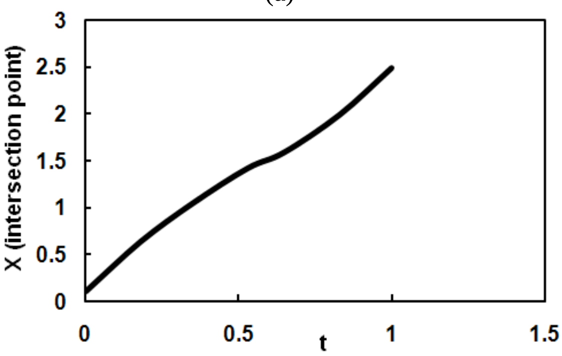
Fig. 14: The secondary water impact.



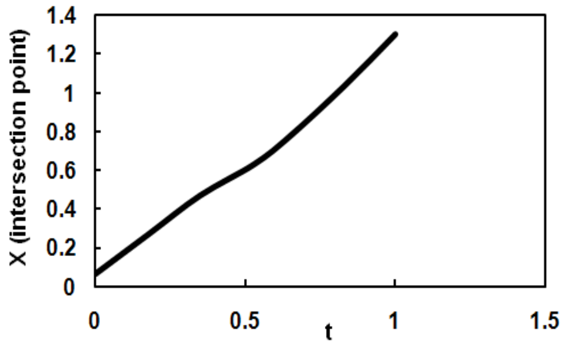
(a)



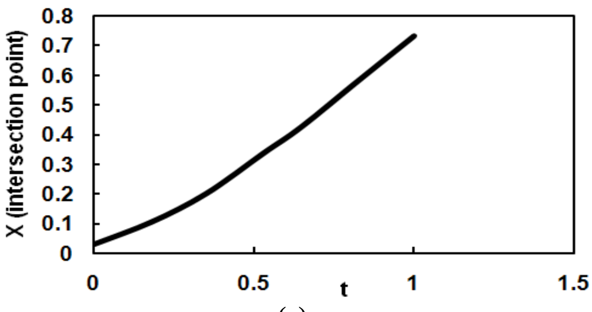
(b)



(c)



(d)



(e)

Fig. 15: The evolution of intersection points; (a) 10 degree, (b) 20 degree, (c) 30 degree, (d) 45 degree, (e) 60 degree

6. Conclusion

In the present study, finite volume method in conjunction with volume of fluid in the ANSYS-CFX solver is implemented to simulate the water entry of arbitrary bow sections. The numerical model has been validated using two different test cases: a 10 degrees wedge section and an arbitrary bow section. The free surface profile and pressure distribution related to some new arbitrary bow sections have been presented and it is observed that by increasing the deadrise angle of the equivalent wedge section, maximum value of pressure decreases. Also, the secondary impact which occurs in some cases has been briefly discussed. It was observed that the secondary impact occurs only at high impact velocities. Finally, an effort has been made to gain better understanding about the flow characteristics near the intersection points of the arbitrary sections.

References

Aarsnes, J.V. (1996): Drop Test with Ship Sections – Effect of Roll Angle, Report 603834.00.01, Norwegian Marine Technology Research Institute, Trondheim, Norway.

Ahmed, Y.M. (2011): Numerical Simulation for the Free Surface Flow Around a Complex Ship Hull Form at Different Froude Numbers", Alexandria Engineering Journal, Vol. 50, No. 3, pp. 229-235. http://dx.doi.org/10.1016/j.aej.2011.01.017

Arai, M., Cheng, L.Y., Inoue, Y., Miyauchi, T. and Ishikawa M. (1995): A Study on Slamming Characteristics and Optimization of Bow Forms of Ships, Proc. 5th Inte. Symp. on Practical Design of Ships and Mobile Units, Seoul, Korea.

Arai, M. and Matsunaga, K. (1989): A Numerical and Experimental Study of Bow Flare Slamming, Journal of Society Naval Architecture Japan. Vol. 166 (in Japanese).

Arai, M. and Tasaki, R. (1987): A Numerical Study of Water Entrance of Two-dimensional Wedges - Effect of Gravity, Spray Generation and Vertical Load, Proc. 3rd Intl. Symp. Practical Design Ships Mobile Units (PRADS'87), Trondheim, Norway.

Battistin, D. and Iafrati, A. (2004): A Numerical Model for the Jet Generation by Water Impact, Journal of Engineering Mathematics, Vol. 48, pp. 353-374. http://dx.doi.org/10.1023/B:engi.0000018173.66342.9f

Battistin, D. and Iafrati,A. (2003): Hydrodynamic Loads During Water Entry of Two-dimensional and Axisymmetric Bodies, Journal of Fluids and Structures, Vol. 17, pp. 643-664. http://dx.doi.org/10.1016/S0889-9746(03)00010-0

Dobrovolskaya, Z.N. (1969): On Some Problems of Similarity Flow of Fluid with a Free Surface, Journal of Fluid Mechanics, Vol. 36, pp: 805–829. http://dx.doi.org/10.1017/S0022112069001996

Farsi, M. and Ghadimi, P. (2014a): Finding the Best Combination of Numerical Schemes for 2-D SPH Simulation of Wedge Water Entry for a Wide Range of Deadrise Angles, International Journal of Naval Architecture and Ocean Engineering, Vol. 6, pp. 638-651.

Farsi, M. and Ghadimi, P. (2014b): Simulation of 2D Symmetry and Asymmetry Wedge Water Entry by Smoothed Particle Hydrodynamics Method, Journal of the Brazilian Society of Mechanical Sciences and Engineering, http://dx.doi.org/10.1007/s40430-014-0212-5.

Farsi, M. and Ghadimi, P. (2014c): Effect of Flat Deck on Catamaran Water Entry through Smoothed Particle Hydrodynamics, Journal of Engineering for the Maritime Environment, http://dx.doi.org/10.1177/1475090214563960.

Ghadimi, P., Dashtimanesh, A. and Feizi Chekab, M. A. (2013): A Numerical Investigation of the Water Impact of an Arbitrary Bow Section, ISH Journal of Hydraulic Engineering, Vol. 19, No. 3, pp. 186-195.

Ghadimi, P., Saadatkhah, A. and Dashtimanesh, A. (2011): Analytical Solution of Wedge Water Entry by Using Schwartz-Christoffel Conformal Mapping, International Journal of Modeling, Simulation and Scientific Computing, Vol. 2 , No.3, pp. 337-354.

Ghadimi, P., Dashtimanesh, A. and Djeddi, S. R. (2012): Study of Water Entry of Circular Cylinder by using Analytical and Numerical Solution, Journal of the Brazilian Society of Mechanical Sciences and Engineering, Vol. 34, No. 3, pp. 225-232. http://dx.doi.org/10.1590/S1678-58782012000300001

Greenhow, M. (1988): Water-entry and -exit of a Horizontal Circular Cylinder, Applied Ocean Research, Vol. 10, pp: 191–8. http://dx.doi.org/10.1016/S0141-1187(88)80003-8

Hargreaves, D.M., Morvan, H.P. and Wright G. (2007): Validation of the Volume of Fluid Method for Free Surface Calculation: the Broad Crested Weir, Engineering Applications of Computational Fluid Mechanics, Vol. 1, No. 2, pp: 136-146.

Kleefsman, K., Fekken, G., Veldman, A., Iwanowski B. and Buchner B. (2005): A Volume-of-Fluid based Simulation Method for Wave Impact Problems, Journal of Computational Physics, Vol. 206, pp. 363–393.

http://dx.doi.org/10.1016/j.jcp.2004.12.007

Kleefsman, T., Loots, E., Veldman, A., Buchner, B., Bunnik, T. and Falkenberg, E. (2005): The Numerical Simulation of Green Water Loading Including the Vessel Motions and the Incoming Wave Field, OMAE Conference, Halkidiki.

Mei, X., Liu, Y. and Yue, D. (1999): On the Water Impact of General Two-dimensional Sections, Appl. Ocean Res. Vol. 21, pp. 1–15. http://dx.doi.org/10.1016/S0141-1187(98)00034-0

Newman, J.N. (1977): Marine Hydrodynamics, The MIT press, Cambridge, MA.

Rahaman, M.M., Akimoto, H. and Ali, M.A. (2013): Numerical Simulation of 2D Hydrodynamic Impact of Wedge and Ship Section at Variable Velocity, Journal of Naval Architecture and Marine Engineering, Vol. 10, pp. 49-58. http://dx.doi.org/10.3329/jname.v10i1.14383

Sun, H. and Faltinsen, O.M. (2006): Water Impact of Horizontal Circular Cylinders and Cylindrical Shells, Applied Ocean Research, 28, pp. 299–311. http://dx.doi.org/10.1016/j.apor.2007.02.002

von Karman, T. (1932): The Impact on Seaplane Float during Landing, NACA TN321, 1929.

Wagner, H. (1932): Über Stoß- und Gleitvorgänge an der Oberfläche von Flüssigkeiten, ZAMM. Vol. 12, pp. 193-215. http://dx.doi.org/10.1002/zamm.19320120402

Yamamoto, Y., Iida, K., Fukasawa, T., Murakami, T., Arai, M. and Ando, A. (1985): Structural Damage Analysis of a Fast Ship due to Bow Flare Slamming, International Shipbuilding Progress, Vol. 32(369), pp. 124-136.

Ying, X., Jorgeson, J. ad, Wang, S.Y. (2009): Modeling Dam Break Flows Using Finite Volume Method on Unstructured Grid, Engineering Application of Computational Fluid Mechanics, Vol. 3, No. 2, pp. 184-194.

Wang, W. and Wang, Y. (2010): An Essential Solution of Water Entry Problems and its Engineering Applications, Journal of Marine Science and Application, Vol. 9, pp. 268-273. http://dx.doi.org/10.1007/s11804-010-1006-5

Zhao, R. and Faltinsen, O.M. (1993): Water Entry of Two-dimensional Bodies, Journal of Fluid Mechanics, Vol. 246, pp. 593–612. http://dx.doi.org/10.1017/S002211209300028X

Zhao, R., Faltinsen, O.M. and Aarsnes, J. (1996): Water Entry of Arbitrary Two-dimensional Sections with and without Separation, Proc. 21st Symposium on Naval Hydrodynamics, pp. 118–133. Trondheim, Norway.

Zhu, X.Y. (2006): Application of the CIP Method to Strongly Nonlinear Wave-body Interaction Problems, Ph.D Thesis, Norwegian University of Science and Technology, Trondheim, Norway.