DEVELOPMENT OF A NUMERICAL HYDRODYNAMIC TANK
FOR SHIP MOTION SIMULATION

R. Panahi*, E. Jahanbakhsh† and M.S. Seif‡

* Marine Laboratory, Department of Mechanical Engineering,
Sharif University of Technology,
P. O. Box 11365-9567 Tehran, I. R. of Iran
e-mail: Roozbeh_Panahi@yahoo.com

Key words: Numerical Hydrodynamics, Volume of Fluid, Fractional Step, Ship Motion

Abstract. In this paper, an integrated computational procedure for simulation of two-phase flow as well as floating and submerged body motions is presented based on a Volume of Fluid (VoF) - fractional step coupling. Two fluids are modeled as a single continuum with a fluid property jump at the interface by solving a scalar transport equation for volume fraction. In conjunction, the conservation equations for mass and momentum are solved using fractional step method. Based on integration of stresses over a body, acting forces and moments are calculated. Using the strategy of non-orthogonal body-attached mesh and calculation of motions in each time step, result in time history of floating or submerged body motions. To verify the accuracy of the numerical procedure in simulation of two-phase flow, sloshing and dam breaking with obstacle problems are investigated. Besides, motions simulation strategy is evaluated by using cylinder water entry test case. To demonstrate the capability of simulation, barge resistance is calculated in two cases of fixed and free motion (2-DoF). All of the results are in good concordance with experimental data. The present method can be extended for full nonlinear motion of ships in waves.

1 INTRODUCTION

Nowadays, numerical simulations are becoming a common way for assessment of ship performance in early design stages. Although model test using experimental approach is still very useful but has its own restrictions and expenses which has motivated to employ numerical simulations. Taking into account the advances in computer hardware, use of Computational Fluid Dynamics (CFD) is becoming the best choice in many cases.

In practice, a ship hydrodynamic problem includes turbulent viscous flow with complex free surface deformation and sometimes fluid-structure interaction. Investigation of the all aspects of such a problem necessitates a robust numerical tool. One of the practical ways to study the aforementioned coupled complicated case is to decouple it by either completely ignoring the less important phenomena or approximating them.

In solving such a problem, one encounters to three subproblems which are: a) velocity and pressure distribution, b) free surface deformation and c) rigid body motion.
Methods for solving the Navier-Stokes and continuity equations are typically categorized as pressure-corrector schemes and projection or fractional step schemes.

In pressure-corrector methods like SIMPLE\(^1\) and PISO\(^2\), a pressure correction equation is solved for several iterations in each time step to reach a divergence free velocity field. In contrast with such iterative methods, in fractional step schemes a pressure or pseudo-pressure Poisson equation is solved once in each time step to enforce continuity. Therefore, using fractional step is preferable, especially in unsteady problems\(^4\). Fractional step method which has been used widely over the past two decades, is introduced by Chorin\(^5,6\), based on Hodge decomposition. In the 1980s, several second-order projection methods are proposed by Goda\(^7\), Bell et al.\(^8\), Kim and Moin\(^9\) and Van Kan\(^10\). Accuracy of such methods is discussed by Brown et al.\(^11\), resulted in introducing a modified scheme proposed by Bell et al.\(^8\).

On the other hand, the existing approaches for handling fluids interface are\(^12\): interface tracking or surface methods and interface capturing or volume methods.

Interface tracking methods are characterized by an explicit representation of the interface. In other words, the computational grid is moved and updated in each time step to have no flow across it while satisfying force equilibrium on fluid at interface\(^4\). The common drawback of this category is the inability to handle complex geometries and overturning waves. This problem leads to interface capturing methods where the interface is captured as a part of the physical domain. Volume of Fluid (VoF) is an approach in volume methods. One of the most interesting schemes in this category is to solve an additional convection equation. This results in volume fraction which implies the availability of two phases in each Control Volume (CV) for whole computational domain. In discretisation of such a transport equation one encounters to face values which must be estimated using an appropriate interpolation scheme. This interpolation must ensure both boundedness and availability criteria\(^13\) to have physical volume fraction values. It means the value of a flow property in the absence of source or sink can not be higher or lower than those values prescribed on the boundaries of a cell. In addition, the amount of flow convected over a face during a time step should be less than or equal to the amount available in donor cell. Simple interpolations have some problem with transitional area between two phases while introducing numerical diffusion\(^14\) or disobeying the local boundedness\(^15\). There are some composite schemes which switch between their options according to the received signals about the flow, to have physical distribution of fluids in whole domain\(^16,17\). CICSAM\(^17\) (Compressive Interface Capturing Scheme for Arbitrary Meshes) which is used in this study is a promising method which appropriately retains the transitional region between two fluids while successfully establishes all criteria in comparison to other composite interpolations\(^18\).

Coupling of two aforementioned subproblems results in simulation of the interfacial flow. This subject is recently developed by many researchers especially based on volume methods\(^19, 20\) and used in simulation of breaking waves\(^21,22\), green water\(^23, 24\) sloshing\(^25\) and wave-structure interaction\(^26\).

Interfacial flow simulation results in velocity and pressure distribution which yield to tangential and normal stresses, respectively. Integration from such stresses over the body results in forces and moments acting on it. Here, the motions of the rigid body in the 6-DoF are determined by solving the equations of variation of linear and angular momentum. This is
the last subproblem which gives a time-history of body motions in one or two phases e.g. submarine maneuvering or ship seakeeping. The important point is that, although potential theory methods are capable of predicting motions with lower run time but they are not suitable where viscous effects or breaking waves play an important role. For several practically important cases large errors can be introduced by the potential theory assumptions.

The motion of a floating body is a direct consequence of the flow-induced forces acting on it while at the same time these forces are a function of the body movement itself. Therefore, the prediction of flow-induced body motion in viscous fluid is a challenging task and requires coupled solution of fluid flow and body motion. In recent two decades, with the change in computer hardware, motion simulation is the subject of many hydrodynamics researches. It is developed from the restricted motions such as trim or sinkage by Miyata, Hochbaum, Alessandrini and Kinoshita to the evaluation of 6-DoF motions by Miyake, Azcueta, Vogt and Xing.

In this paper an integrated finite volume procedure for simulation of three-dimensional interfacial flow interaction with rigid body motion is presented. The governing equations are reviewed in section 2.1. Discretisation of the pressure integral in Navier-Stokes equations is of great importance especially when there are two phases with high density ratio e.g. water and air. Discretisation of the governing equations are presented in section 2.2 besides the introduction of a new pressure integral discretisation. Special care must be taken in motion simulation about the Reference System (RS) selection and its effect in governing equations which is applied everywhere needed.

A computer software is developed based on the presented mathematical formulation. Interfacial flow part of the software is verified using sloshing and dam breaking with obstacle problems. The strategy of motion simulation is assessed with simulation of cylinder water entry problem. Finally, barge resistance is calculated in fixed and free motion (2-DoF) cases. Comparisons of all test cases show the ability of the proposed procedure to be extended as a numerical hydrodynamic tank for full nonlinear ship motion simulation.

2 NUMERICAL METHOD

2.1 Governing Equations

As mentioned earlier, complex deformation of interface in hydrodynamic problems leads to use of volume methods in free surface modeling. Here a scalar transport equation which is extracted from the continuity equation (Eq.1) is solved to determine the volume fraction of each phase (i.e. air and water) in all computational cells:

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot (\alpha \bar{u}) = 0$$

(1)

This results in $\alpha$ distribution as:

$$\alpha = \begin{cases} 
1 & \text{for cells inside fluid 1} \\
0 & \text{for cells inside fluid 2} \\
0 < \alpha_0 < 1 & \text{for transitional area}
\end{cases}$$

(2)
Here, an effective fluid with variable physical properties is introduced:

$$\rho_{\text{eff}} = \alpha \rho_1 + (1 - \alpha) \rho_2$$  \hspace{1cm} (3)

$$\nu_{\text{eff}} = \alpha \nu_1 + (1 - \alpha) \nu_2$$

Where subscripts 1 and 2 represent two phases e.g. water and air.

Such an effective fluid is used in solving the fluid main governing equations. These are momentum and continuity equations which are as follow:

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial P}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_i \partial x_j} + g_i$$ \hspace{1cm} (4)

$$\frac{\partial u_i}{\partial x_i} = 0$$ \hspace{1cm} (5)

Velocity and pressure distribution around a body causes it to move in 6-DoF. These movements can be estimated by solving the linear and angular momentum equations:

$$\sum F = m\ddot{a}$$ \hspace{1cm} (6)

$$\sum \dot{M}_G = I_G \ddot{\alpha} + \dddot{\alpha} \times I_G \ddot{\omega}$$ \hspace{1cm} (7)

Finally, it must be mentioned that, a body-attached mesh which follows the time history of body motions is used for rigid body motion simulation. In addition, all of the fluid governing equations are written for a moving Control Volume (CV) in Newtonian RS (Figure 1).

2.2 Discretisation

The finite volume discretisation of volume fraction transport equation (Eq.1) is based on the integration over the CV and the time step:

$$\int_{t}^{t+\delta t} \left( \int_V \frac{\partial \alpha}{\partial t} dV \right) dt + \int_{t}^{t+\delta t} \left( \int_V \nabla \cdot (\alpha \vec{u}) \right) dV \right) dt = 0$$ \hspace{1cm} (8)
The first term in Eq.8 is a common integral form and applying the Gauss theorem on the second term results in:

\[
(a_t^{t+\delta t} - a_t^t) \frac{V}{\Delta t} + \frac{1}{2} \left( \sum_{f=1}^{n} a_f^{t+\delta t} F_{f-rel}^{t+\delta t} + \sum_{f=1}^{n} a_f^t F_{f-rel}^t \right) = 0
\]  

(9)

Where \( a_f \) is the face volume fraction. \( F_{f-rel} = (\tilde{U}_f - \tilde{U}_m) \). \( \tilde{A}_f \) is the relative volumetric flux at the CV face. \( \tilde{U}_f \) is calculated in an especial manner and will be discussed later in the solution algorithm (section 2.3.1). In addition, \( \tilde{U}_m \) is the only effect of moving CV in governing equations.

The time integral of the second term, is discretised using Crank-Nicholson scheme. Assuming a linear and small variation of \( F_{f-rel} \) in the small time step, results in using the most recent value of it. Therefore, rearranging Eq.9 leads to:

\[
\left( a_t^{t+\delta t} - a_t^t \right) \frac{V}{\Delta t} + \frac{1}{2} \sum_{f=1}^{n} a_f^{t+\delta t} F_{f-rel}^{t+\delta t} = S_{\alpha_p}
\]  

(10)

Where \( \Delta t \) is the time step and the source term \( S_{\alpha_p} \) in Eq.10 is written as follows:

\[
S_{\alpha_p} = a_t^t \frac{V}{\Delta t} - \sum_{f=1}^{n} \frac{1}{2} a_f^t F_{f-rel}
\]  

(11)

\( a_f \) in Eqs.10 and 11 must be approximated based on CVs’ center values. It has been mentioned earlier that, simple interpolations result in non-physical volume fraction values. This leads to using a high order composite interpolation.

Most of composite interpolations, typically switch between two high and low order interpolations in order to use their advantages and minimize their disadvantages. Here, the main distinctions are that how and when they switch between these schemes according to flow information.

CICSAM (Compressive Interface Capturing Scheme for Arbitrary Meshes) uses CBC\(^{37}\) (Convection Boundedness Criteria) (\( \tilde{a}_{f-CBC} \)) and UQ\(^{14}\) (ULTIMATE QUICKEST) (\( \tilde{a}_{f-UQ} \)) interpolations by introducing a weighting factor \( \gamma_f \) (Eq.12) which takes into account the slope of the free surface relative to the direction of motion. CBC is the most compressive scheme which stipulates robust local bounds on \( \tilde{a}_f \) nevertheless does not actually preserve the shape of interface. Here UQ is used for its ability in preserving of interface shape. Based on NVD\(^{14}\) (Normalized Variable Diagram) normal face value is obtained as follow:

\[
\tilde{a}_f = \gamma_f \tilde{a}_{f-CBC} + (1 - \gamma_f) \tilde{a}_{f-UQ}
\]  

(12)
NVD definitions are shown in Figure 2 where donor, acceptor and upwind cells (subscripts $D$, $A$ and $U$ respectively) are defined according to flow direction for each CV face:

$$\tilde{\alpha}_f = \frac{\alpha_f - \alpha_U}{\alpha_A - \alpha_U}$$  (13)

Using the definition of Eq.13 in Eq.12 results in estimation of $\alpha_f$.

$\alpha_f$ contains all information regarding to the fluid distribution in the doner, acceptor and upwind cells as well as the interface orientation relative to flow direction. To avoid the non-physical $\alpha$, a correction step is added to volume fraction calculation procedure and used in the developed software which can be found in Ubbink and Issa by details.

By integration of the Navier-Stokes equations over a moving CV it becomes as Eq.14 in Newtonian RS:

$$\frac{d}{dt} \int V \, dV + \int A \left( \bar{u} \cdot \bar{n} \right) \, dA = \int A \left( \nabla \bar{u} \cdot \bar{n} \right) \, dA - \frac{1}{\rho} \int V \nabla P \, dV + \int V \, g \, dV$$  (14)

For the fluid velocity $u_i$ the diffusion term (the first term in r.h.s. of Eq.14) is discretised using over-relaxed interpolation:

$$\int A \left( \nabla \bar{u}_i \cdot \bar{n} \right) \, dA = \sum_{faces} \bar{A}_f \left( \nabla \bar{u}_i \right)_f$$  (15)

In order to discretise the convection term (the second term in l.h.s. of Eq.14), one needs fluid velocity at CV face $u_{i-f}$ as well as the volumetric flux which are appeared in Eq.16:

$$\int A \left( \bar{u}_{Rel} \cdot \bar{n} \right) \, dA = \sum_{faces} u_{i-f} F_{f-Rel}$$  (16)

Where $u_{i-f}$ is calculated using Gamma interpolation scheme.

It must be mentioned that the Crank-Nicholson scheme is used for time discretisation of diffusion and convection terms in Eq.14.
The pressure term (second term in r.h.s. of Eq.14) is discretised as:

$$\int_P \bar{n} dA = \sum_{face} P_f \bar{A}_f$$  \hspace{1cm} (17)$$

Using the common Linear Interpolations (LI) for calculation of face pressure $P_f$, results in severe oscillations in velocity field. This is of great importance especially when there are two fluids with high density ratio e.g. water and air. Here a Piecewise Linear Interpolation (PLI) is introduced as shown in Figure 3. It is based on a constraint for lines $L_{Af}$ and $L_{Bf}$ which connect pressure values at CVs’ center $P_A$ and $P_B$ to $P_f$ as follow:

$$\frac{\text{Slope of } L_{Af}}{\text{Slope of } L_{Bf}} = \frac{\rho_A}{\rho_B}$$ \hspace{1cm} (18)$$

Where $\rho_A$ and $\rho_B$ are the densities of CVs A and B, respectively. Therefore $P_f$ can be estimated by using the pressure value at CVs’ center $P_A$ and $P_B$ by Eq.19:

$$P_f = P_A \kappa + P_B (1 - \kappa)$$ \hspace{1cm} (19)$$

Where the weighting factor $\kappa$ is calculated as follows:

$$\kappa = \frac{\rho_B \delta_B}{\rho_A \delta_A + \rho_B \delta_B}$$ \hspace{1cm} (20)$$

In this equation $\delta_A$ and $\delta_B$ are distance from face center $f$ to CVs’ center $A$ and $B$, respectively.

In order to show the effect of such an interpolation, the hydrostatic pressure distribution along the near free surface cells (Figure 4-a) is plotted in Figure 4-b. In Fig 4-a points and solid lines stand for CVs’ centers and faces, respectively.

Figure 5 shows the pressure distribution at CVs’ faces calculated by using LI and PLI. It is obvious from Figure 5 that using LI results in non-physical pressure at CV’s face, but PLI calculates the CV’s face pressure exactly which is zero for the face at $x = 0$ m. This difference affects on pressure integral of each CV consequently, as shown in Figure 6. Although it seems that such a difference by using LI in comparison to PLI can be neglected, but dividing of pressure integral term by CV’s density (Eq.6) exaggerates this error. Especially in the case of large density ratio of two phases and for the CV next to the free surface in the light fluid (here at $x=0.5$ m) it is very important. Therefore using PLI restrains severe oscillations of the velocity field by better estimation of $P_f$. 

Figure 3: PLI for CV’s face pressure calculation

Figure 4: Hydrostatic pressure distribution at near free surface CVs’ center

Figure 5: Pressure at CV’s centers and faces

Figure 6: Pressure integral
2.3 Solution algorithm
As aforementioned, one encounters to three problems through the simulation of a hydrodynamics motion which are shown in Figure 7 with their connections.

2.3.1 Velocity and pressure distribution
In solving the Navier-Stokes and continuity equations one needs physical properties -density and viscosity distribution- in the computational domain. Solving the volume fraction transport equation (Eqs.10 and 11) using the CICSAM interpolation results in an effective fluid properties using Eq.3. For calculation of velocity and pressure fields, the fractional step method of Kim and Choi\textsuperscript{39} is applied as shown in Figure 8.

The first step is to solve the momentum equation with the lagged pressure gradient (Eq.21) and then to calculate the intermediate velocity $\mathbf{u}^i$ by Eq.25. Using Eq.26 results in new pressure field $P^{n+1}$. Especial care must be taken into account for calculation of the r.h.s. integral of Eq.26 in boundary faces. In other words, the fluid velocity at face $\mathbf{U}_{f}^{n}$ calculated in the previous time step, must be used at the boundary face rather than the intermediate one. Also the intermediate CV’s face velocity $\mathbf{U}_{f}^{i}$ is calculated using the linear interpolation.

Eqs.27 and 28 are used in calculation of fluid velocity at CV’s center and CV’s face, respectively. It must be noted that in the colocated arrangement used in the current algorithm, face velocities must be calculated separately including the effect of pressure gradient to overcome to checkerboard pressure and Eq.28 is used for such a purpose\textsuperscript{40}.
2.3.2 Motions simulation

Solving the Navier-Stokes and continuity equations for an effective fluid result in flow forces $\vec{F}_{\text{flow}}$ and moments $\vec{M}_{G_{\text{flow}}}$ which are calculated by integrating the pressure field and viscous stresses over the body as Eqs.29 and 30:

$$\vec{F} = \vec{F}_{\text{ext}} + \vec{W} + \vec{F}_{\text{flow}} + \bar{m} \bar{g} + \sum_{j=1}^{n} ( - P_j \bar{n}_j + \bar{r}_j ) \hat{A}_j$$

$$\vec{M}_G = \vec{M}_{G_{\text{ext}}} + \vec{M}_{G_{\text{flow}}} = \sum_{j=1}^{n} ( \bar{F}_j - \bar{r}_j ) \times ( - P_j \bar{n}_j + \bar{r}_j ) \hat{A}_j$$

Where $\vec{F}$ is the total force in the Newtonian RS and $\vec{M}_G$ is the total moment around the mass center. $\vec{F}_{\text{ext}}$ and $\vec{M}_{G_{\text{ext}}}$ can be any external forces and moments acting on the body in the Newtonian RS. One can use them to simulate propeller or rudder, towing, sailing, etc. $\vec{W} = m \bar{g}$ is the body weight force which has only one component along Z-direction in Newtonian RS. Subscript $j$ stands for each CV’s face defining the body surface. Also in Eq.30 $\bar{r}_j$ is the position vector of the CV’s face center for all faces defining the body and $\bar{r}_G$ is the...
position vector of the body mass center. The important point is that, according to the selected RS, all vectors must be in the Newtonian RS.

Solving the Eqs. 6 and 7 with calculated forces and moments based on Crank-Nicholson time discretisation results in body motions which are applied on the body-attached mesh in each time step. Such a procedure makes ready the physical geometry for the next time step.

3 Numerical Results

In order to assess the feasibility, computer software is developed according to above mentioned procedure. This is summarized in some conclusion about the accuracy, efficiency and robustness of two parts which are two phase flow and hydrodynamic motion simulation.

3.1 Sloshing Problem

To test the interfacial flow solver, sloshing of a liquid wave with low amplitude under the influence of gravity is investigated. The considered situation is shown in Figure 9 which is the same as used by Raad et al.\(^4\). Initially the quiescent fluid has an average depth of 0.05 m, and its surface is defined by \(y(x) = 0.05 + 0.005\cos (\pi x)\). The computational domain used is an open container with a base length of 0.1 m and a height of 0.065 m (Figure 9). Also, its bottom and sides are treated as slip boundaries. The domain is discretised with 160 cells in the horizontal direction and 104 cells in the vertical direction. In this case, the fluid begins to slosh solely under the influence of constant gravitational field. The theoretical period of sloshing of the first mode is\(^4\):

\[
P = 2\pi \sqrt{gk \tanh (kh)} = 0.3739 s
\]

(31)

Where \(k\) is the wave number and \(h\) the average fluid length.

Figure 9: Initial geometry of the sloshing problem

Initially the whole system is at rest. After a quarter of a period the potential energy of the system has been transferred to kinetic energy and the velocities reach their maximum. After a half period, all the kinetic energy has been transferred back into potential energy with the
velocity almost back to zero (Figure 10). Also, Figure 11 shows plots of the position of the interface at the left boundary against time. The frequency corresponds with the theoretical one, so do the amplitudes of even periods.

Figure 10: Plots of the wave position for the first period of the sloshing

Figure 11: Position of the interface at the left boundary

3.2 Dam Breaking with Obstacle

A more interesting version of dam breaking occurs when a small obstacle is placed in the way of the water front as shown in Figure 12. In this case, considering the flow of both water and air is important because air is trapped in water; due to its much lower density, trapped-air is subjected to a large buoyancy force and tends to rise up. This is obvious from Figure 13 which shows the shape of a water column at three time instants. Comparison of numerical and experimental photos of Figure 13 shows a good concordance.
3.3 Water entry of a horizontal circular cylinder

To evaluate the rigid body motion coupling with fluid flow, water entry of a circular cylinder is studied. The neutrally-buoyant circular cylinder and the computational grid of 18900 CVs is considered as shown in Figure 14. The cylinder is released from a position just above the still water level. It intersects the water surface with the downward velocity of 4 m/s. Here, no-slip boundary condition at cylinder wall, zero value at down boundary and zero-gradient at other boundaries are applied on velocity. Also, zero-gradient condition is used for pressure at whole boundaries.
After the cylinder impacts on water surface, the velocity of cylinder is decreased significantly due to the effects of hydrodynamic forces. As shown in Figure 15 for three time instants, water spray is thrown up at each side of the cylinder and travel straight upward until they become unstable. Figure 16 shows the time history of vertical displacement and acceleration of the cylinder. The instantaneous vertical positions of the cylinder are compared with experimental data of Greenhow\textsuperscript{43} and numerical simulation of Xing\textsuperscript{35}. It shows a reasonably good agreement with experimental data in comparison to similar numerical study.
3.4 Barge resistance

Ship resistance is usually evaluated with fixed trim and draft, but they may change in moving conditions due to hydrodynamic effects. In order to show the importance of motion simulation—especially in ship resistance calculation—a barge ship is simulated in two cases of fixed and 2-DoF motion and the numerical results are compared with the experimental data. The barge model characteristics are shown in Figure 17 and Table 1. Besides, the computational grid of 36000 cells is shown in Figure 18. No-slip and zero-gradient boundary conditions are applied for velocity on wall and other boundaries, respectively. Also, zero-gradient boundary condition is used for pressure in all boundaries. In order to minimize the reflection of flow a damping zone is considered in outlet boundaries.

Experimental test is done at $V=0.807 \text{ m/s}$ in marine laboratory of Shrif University of Technology. It is obvious that the barge ship is not a streamline body and therefore the wave making resistance component (deformation of free surface) of total resistance is of great importance in comparison to viscous resistance component. Figure 19 shows the free surface deformation in front of barge which is of good concordance with experiment. This results in appropriate prediction of total resistance although it could be captured better with finer mesh.

![Figure 16](image1.png)

**Figure 16:** Time history of the vertical motion (left) and the impact force (right) during the cylinder water entry

<table>
<thead>
<tr>
<th>Acc/Grav</th>
<th>20</th>
</tr>
</thead>
<tbody>
<tr>
<td>16</td>
<td>12</td>
</tr>
<tr>
<td>8</td>
<td>4</td>
</tr>
<tr>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Acc/Grav</th>
<th>0.05</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.1</td>
<td>0.15</td>
</tr>
<tr>
<td>0.2</td>
<td>0.05</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>time(s)</th>
<th>0</th>
<th>0.03</th>
<th>0.06</th>
<th>0.09</th>
<th>0.12</th>
<th>0.15</th>
<th>0.18</th>
<th>0.21</th>
<th>0.24</th>
</tr>
</thead>
<tbody>
<tr>
<td>y(m)</td>
<td>-0.05</td>
<td>-0.1</td>
<td>-0.15</td>
<td>-0.2</td>
<td>-0.25</td>
<td>-0.3</td>
<td>-0.35</td>
<td>-0.4</td>
<td>-0.45</td>
</tr>
</tbody>
</table>

**Figure 17:** Barge ship geometry definitions

**Table 1:** Barge Ship characteristics

<table>
<thead>
<tr>
<th>L</th>
<th>1.05 m</th>
</tr>
</thead>
<tbody>
<tr>
<td>B</td>
<td>0.29 m</td>
</tr>
<tr>
<td>T</td>
<td>0.025 m</td>
</tr>
<tr>
<td>$C_a$</td>
<td>1.0</td>
</tr>
<tr>
<td>Mass</td>
<td>7.26 Kg</td>
</tr>
<tr>
<td>$I_{yy}$</td>
<td>0.7 Kg m²</td>
</tr>
<tr>
<td>KG</td>
<td>0.025 m</td>
</tr>
</tbody>
</table>
In the case of 2-DoF motion, barge is free to heave and pitch and the resistance is calculated when two aforementioned motions are approximately constant. It must be noted that, many factors included in the accuracy of resistance calculation as well as turbulence flow and grid
resolution are same in fixed and 2-DoF motion. Taking this into account, the error is decreased from 23.2% in fixed motion into 5.9% in 2-DoF motion which is similar to test conditions as shown in Table 2.

In other words, since the model test is performed with free model (heave and pitch motions were not restricted), the numerical results in 2-DoF motion simulation is much better than fixed condition. This means that in usual numerical resistance we always have errors if we can not predict the moving condition (draft and trim) with good accuracy. But the present numerical method can find out moving conditions automatically.

<table>
<thead>
<tr>
<th>Resistance</th>
<th>Value (N)</th>
<th>Error (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experiment Calculation</td>
<td>3.53 (N)</td>
<td>-</td>
</tr>
<tr>
<td>Numerical calculation in fixed motion</td>
<td>2.71 (N)</td>
<td>23.2%</td>
</tr>
<tr>
<td>Numerical calculation in 2-DoF motion</td>
<td>3.32 (N)</td>
<td>5.9%</td>
</tr>
</tbody>
</table>

Table 2: Barge ship resistance

### 4 CONCLUSIONS

Basic mathematical formulation for motion simulation with or without free surface is explained and details of numerical discretisation procedures are presented. The method may be applied for prediction of any 3D hydrodynamic motion as well as ship and submarine simulation. A computer software is developed based on the derived method and validated with different numerical simulations such as sloshing and slamming. In all cases very good agreement with available data are achieved. Therefore the software may be used as a 3D numerical hydrodynamic tank. As the first step, a barge ship in steady forward motion is simulated and its resistance is evaluated with 2DoF. Results show the possibility for fully nonlinear simulation of ship motions.

### REFERENCES


