Toward wave-body interaction roblems using CIP method: A demonstrating 2 phase problem

- **Tran Tien Anh**
- **Bui Quan Hung**

Ho Chi Minh city University of Technology, VNU-HCM

(Manuscript Received on July 08th, 2015, Manuscript Revised September 23rd, 2015)

ABSTRACT:

CIP (constrained interpolation profile) is one of the CFD (computational fluid dynamics) methods developed by Japanese professor Takashi Yabe. It is used to simulate 3 phase problems including air on the surface, liquid and structure in solid form. To check the validity of CIP theory, experiments with different problems have been implemented and obtained very positive results. This proves the correctness of the CIP method.

Based on the need of simulation of wave structure interaction (water wave with float of seaplanes, wing in ground effect crafts, piers, drilling, casing ships...), this paper applies the theory of CIP method to find the answer to the problem of 2D simulation via a obstacle. Objectives to do are understanding the physics, finding out the differential equations describing the phenomenon, then proceeding discrete, setting up algorithms and finding out solution of the equations. This paper uses Matlab software to write programs and displays the results.

Key words: *numerical algorithm, constrained interpolation profile, free surface problem, fluid structure interaction, multiphase flows, governing equations.*

1. INTRODUCTION

1.1.Objectives

It is very important to know interaction of water waves on structures (body and float of seaplanes, flying boats, piers, drilling, casing ships...). The main objective of this paper is to establish a numerical prediction way for how water waves impact to a solid body.

Purpose of this paper includes constructing algorithms and computational simulation modules, calculating the fluid forces acting on the structure (lift, drag, torque) and processing and displaying calculated results.

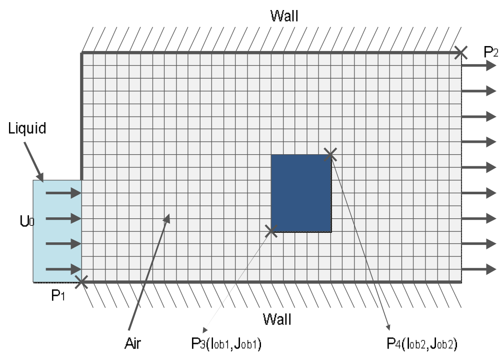

Figure 1. Two phases flow (initial frame).

1.2. Missions

CIP is a CFD method developed by a Japanese professor [1]. CIP is used to simulate 3-phase environments consisting of air over the surface, liquid and a structure. The problem can be understood simply as follows:

- Using equations to describe the movement of water waves.
- Discretizing mathematical equations to establish algorithms programmed on the computer to find the answer.
- Using the programming language to calculate an explanation of the equations.
- Using graphics software to display the results of the problem found in graphs image. Software used in this paper is Matlab.

2. GOVERNING EQUATIONS [1]

From the basic conservation equations:

$$\frac{\partial p}{\partial t} + u_i \frac{\partial p}{\partial x_i} = -\rho C^2 \frac{\partial u_i}{\partial x_i} \qquad (1)$$

Where

t is the time variable;

x_i (i =1,2) are the coordinates of a Cartesian coordinate system;

ρ is the mass density;

u_i (i=1,2) are the velocity components;

f_i (i=1,2) are due to the gravityorce.

$$\sigma_{ij} = -p\delta_{ij} + 2\mu\left(1 - \delta_{ij}/3\right)S_{ij} \qquad (2)$$

where:

σ_{ij} is the total stress

p is the pressure;

μ is the dynamic viscosity coefficient;

δ_{ij} is the Kronecker delta function;

$$S_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \qquad (3)$$

Kronecker delta function:

$$\delta_{ij} = \begin{cases} 0 \text{ if } i \neq j \\ 1 \text{ if } i = j \end{cases}$$

C is sound speed.

In order to identify which part is the air, the water or the solid body, density functions φm (m=1, 2, 3) is introduced:

$$\varphi_m(x,y,t) = \begin{cases} 1, & (x,y) \in \Omega_m \\ 0, & \text{otherwise} \end{cases}$$

where Ω_m : domain occupied by the liquid, gas and solid phase, respectively.

These functions satisfy:

$$\frac{\partial \varphi_m}{\partial t} + u_i \frac{\partial \varphi_m}{\partial x_i} = 0 \qquad (4)$$

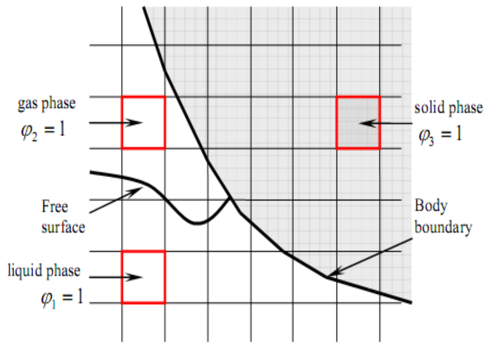

Figure 2. Density function ϕm (m=1,2,3) for multiphase problems with 0≤ ϕm ≤ 1 and ϕ1 + ϕ2 + ϕ3 = 1 in the computational cells.

3. CIP METHOD

3.1. Principle of CIP Method [2]

CIP method has some advantages over other methods with respect to the treatment of advection terms. In this section, the principle of CIP method is explained. Figure 3 shows the principle of CIP method. Here, a one-dimensional advection equation is used to simplify the explanation of CIP method. As mentioned in the previous section, a one-dimensional advection equation is described as below,

$$\frac{\partial f}{\partial t} + u\frac{\partial f}{\partial x} = 0 \qquad (5)$$

The approximate solution of the above equation is given as:

$$f\left(x_i, t + \Delta t\right) \cong f\left(x_i - u\Delta t, t\right)$$

Where x_i is the coordinates of calculation grid. The above equation indicates that a specific profile of f at time $t + \Delta t$ is obtained by shifting the profile at time t with a distance $u\Delta t$ as shown in Figure 3(a). In the numerical simulation, however, only the values at grid points can be obtained, as shown in Figure 3(b). If we eliminate the dashed line shown in Figure 3 (a), it is difficult to imagine the original profile and is naturally to retrieve the original profile depicted by solid line in (c). This process is called as the first order upwind scheme [3]. On the other hand, the use of quadratic interpolation, which is called as Lax-Wendroff scheme [4] or Leith scheme [5], suffers from overshooting.

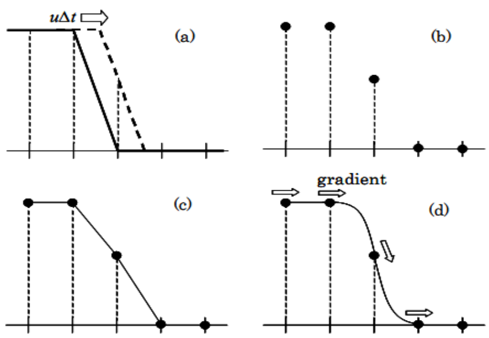

Figure 3. The principle of CIP method: (a) solid line is initial profile and dashed line is an exact solution after advection, whose solution (b) at discretized points, (c) when (b) is linearly interpolated, and (d) In CIP [6]

In CIP method, a spatial profile within each cell is interpolated by a cubic polynomial. Differentiating equation (5) with a spatial variable x gives:

$$\frac{\partial g}{\partial t} + u\frac{\partial g}{\partial x} = -g\frac{\partial u}{\partial x} \qquad (6)$$

By this equation the time evolution of f and g can be traced on the basis of Equation (5). If g propagates in the way shown by the arrows in Figure 3(d), the profile looks smoother that is more precise. It is not difficult to imagine that by this treatment, the solution becomes much closer to the original profile. If two values of f and g are given at two grid points, the profile between the points can be described by a cubic polynomial:

$$F\left(x\right) = ax^3 + bx^2 + cx + d$$

The profile at n+1 step can be obtained by shifting the profile with $u\Delta t$,

$$f^{n+1} = F\left(x - u\Delta t\right)$$

$$g^{n+1} = \frac{\partial F\left(x - u\Delta t\right)}{\partial x} \qquad (7)$$

3.2. Separation of Equations

The governing equations of the fluid and the density function is:

$$\frac{\partial}{\partial t}\begin{pmatrix}\rho \\ u_j \\ p \\ \varphi_m\end{pmatrix} + u_i\frac{\partial}{\partial x_i}\begin{pmatrix}\rho \\ u_j \\ p \\ \varphi_m\end{pmatrix} = \begin{pmatrix}0 \\ 0 \\ 0 \\ 0\end{pmatrix} + \begin{pmatrix}0 \\ \frac{2\mu}{\rho}\frac{\partial}{\partial x_j}\left(S_{ij} - \frac{1}{3}\delta_{ij}S_{kk}\right) + f_j \\ 0 \\ 0\end{pmatrix} + \begin{pmatrix}-\rho\frac{\partial u_i}{\partial x_i} \\ -\frac{1}{\rho}\frac{\partial p}{\partial x_i} \\ -\rho C^2\frac{\partial u_i}{\partial x_i} \\ 0\end{pmatrix} \qquad (8)$$

This equation is separated into three parts

Advection phase:

$$\frac{\partial}{\partial t}\begin{pmatrix}\rho\\ u_j\\ p\\ \varphi_m\end{pmatrix}+u_i\frac{\partial}{\partial x_i}\begin{pmatrix}\rho\\ u_j\\ p\\ \varphi_m\end{pmatrix}=\begin{pmatrix}0\\0\\0\\0\end{pmatrix} \tag{9}$$

Non-advection phase 1:

$$\frac{\partial}{\partial t}\begin{pmatrix}\rho\\ u_j\\ p\\ \varphi_m\end{pmatrix}=\begin{pmatrix}0\\ \frac{2\mu}{\rho}\frac{\partial}{\partial x_j}\left(S_{ij}-\frac{1}{3}\delta_{ij}S_{kk}\right)+f_j\\ 0\\ 0\end{pmatrix} \tag{10}$$

Non-advection phase 2:

$$\frac{\partial}{\partial t}\begin{pmatrix}\rho\\ u_j\\ p\\ \varphi_m\end{pmatrix}=\begin{pmatrix}-\rho\frac{\partial u_i}{\partial x_i}\\ -\frac{1}{\rho}\frac{\partial p}{\partial x_i}\\ -\rho C^2\frac{\partial u_i}{\partial x_i}\\ 0\end{pmatrix} \tag{11}$$

Instead of calculating $f^n \rightarrow f^{n+1}$ (n is time step) directly from Equation (7), intermediate value of f^* and f^{**} are provided, and $f^n \rightarrow f^*$ using Equation (9), $f^* \rightarrow f^{**}$ using Equation (10), $f^{**} \rightarrow f^{n+1}$ using Equation (11) are calculated.

After obtained components of velocity, density, pressure, function of density; spatial derivatives of these components, $\left(\frac{\partial f}{\partial x}\right), \left(\frac{\partial f}{\partial y}\right)$, can be calculated.

This procedure can be summarized as Table 1.

Table 3. Procedure of separation solution

Advection phase	$f^n \rightarrow f^*$ $(\partial_x f)^n \rightarrow (\partial_x f)^* \rightarrow \overline{(\partial_x f)}$ $(\partial_y f)^n \rightarrow (\partial_y f)^* \rightarrow \overline{(\partial_y f)}$
Non-advection phase 1	$f^* \rightarrow f^{**}$ $\overline{(\partial_x f)} \rightarrow (\partial_x f)^{**}$ $\overline{(\partial_y f)} \rightarrow (\partial_y f)^{**}$
Non-advection phase 2	$f^{**} \rightarrow f^{n+1}$ $(\partial_x f)^{**} \rightarrow (\partial_x f)^{n+1}$ $(\partial_y f)^{**} \rightarrow (\partial_y f)^{n+1}$

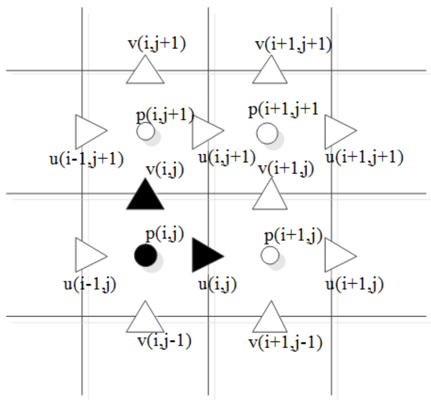

Figure 4. Computational grid distributions

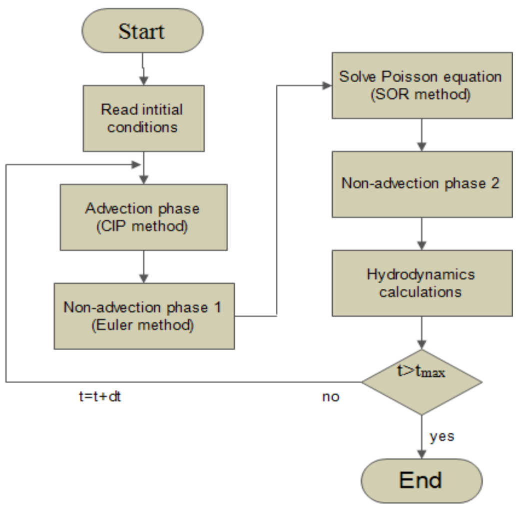

Figure 5. Computational procedures

4. NUMERICAL SIMULATION

4.1. Problem Statement

Two-dimensional water interacting with a solid body is considered in this section. The fluid is assumed to be incompressible and inviscid. Temperature variations are neglected. The problem is described in Figure 6.

2-phase problem is the first step, the base premise to write programs for 3-phase problem and absolutely no experimental verification` [5].

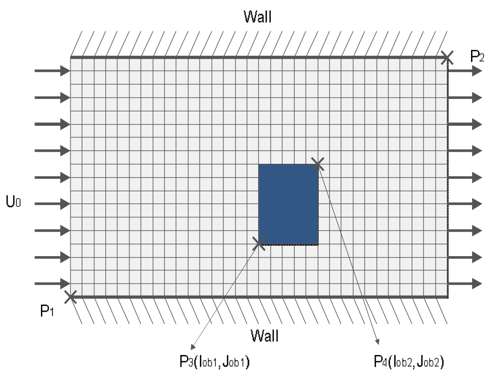

Figure 6. Two phases flow (initial frame)

In which,

U_0 is inlet velocity.

Computational domain is presented by two points P_1 and P_2.

Obstacle is presented by two points P_3 and P_4, as shown in Figure 6.

4.2. Boundary Grid Structure

Boundary grid structure is shown in Figure 7, 8 and 9.

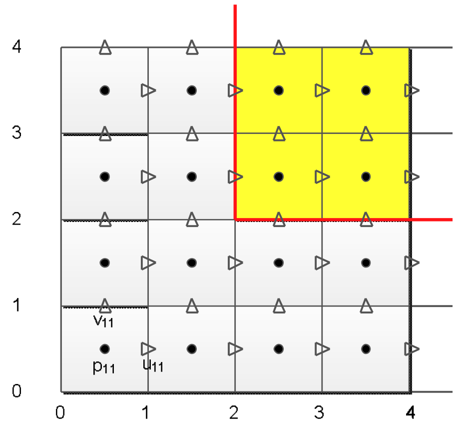

Figure 7. Boundary grid structure (left-bottom)

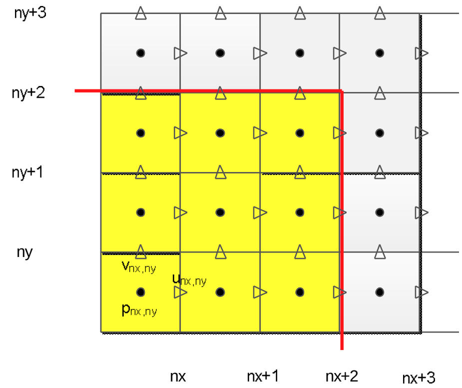

Figure 8. Boundary grid structure (right-top)

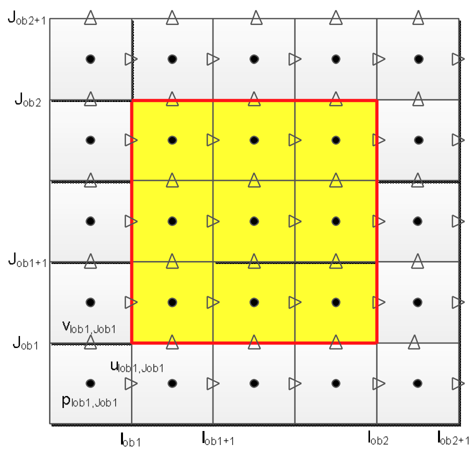

Figure 9. Boundary grid structure (obstacle)

4.3. Boundary Conditions

Inlet boundary condition:

$$u_{2,3:ny+2} = U_0$$

$$v_{2,3:ny+2} = 0$$

Outlet boundary condition:

$$u_{nx+2,3:ny+2} = 2 \times u_{nx+1,3:ny+2} - u_{nx,3:ny+2}$$

$$v_{nx+3,3:ny+1} = v_{nx+2,3:ny+1}$$

Bottom wall boundary condition:

$$u_{2:nx+2,2} = -u_{2:nx+2,3}$$

$$v_{2:nx+3,2} = 0$$

Top wall boundary condition:

$$u_{2:nx+2,ny+3} = -u_{2:nx+2,ny+2}$$

$$v_{2:nx+3,ny+2} = 0$$

Condition for obstacle

$$u_{Iob1:Iob2,Job1+1:Job2} = 0$$

$$v_{Iob1+1,Job1+1:Job2-1} = -v_{Iob1,Job1+1:Job2-1}$$

$$v_{Iob2,Job1+1:Job2-1} = -v_{Iob2+1,Job1+1:Job2-1}$$

$$v_{Iob1+1:Iob2,Job1:Job2} = 0$$

$$u_{Iob1+1:Iob2-1,Job1+1} = -u_{Iob1+1:Iob2-1,Job1}$$

$$u_{Iob1+1:Iob2-1,Job2} = -u_{Iob1+1:Iob2-1,Job2+1}$$

4.4. Boundary Condition for Poisson's Equation

Inlet boundary condition:

$$p_{2,3:ny+2} = p_{3,3:ny+2}$$

Bottom wall boundary condition:

$$p_{2:nx+2,2} = p_{2:nx+2,3}$$

Top wall boundary condition:

$$p_{2:nx+2,ny+3} = p_{2:nx+2,ny+2}$$

5. RESULTS

The computing Matlab program was developed to perform this problem. In this program:

Computational domain (m): $P_1(x_1,y_1)$ and $P_2(x_2,y_2)$.

Obstacle position (m): $P_3(x_3,y_3)$ and $P_4(x_4,y_4)$.

Coordinate of obstacle: P_3(Iob1, Job1) and P_4(Iob2, Job2).

Number of mesh in two axis x, y are: n_x and n_y respectively.

The size of a small grid is : h (h = Δx=Δy) .

Time step : dt.

Number of time step: n_t

Inlet velocity: U_0.

With:

U_0 =10 (m/s),	d_t=0.002
x_1=0,	y_1=0,
x_2=0.02,	y_2=0.01,
x_3=0.45*x_2,	y_3=0.1*y_2;
x_4=0.6*x_2,	y_4=0.65*y_2;

The velocity vector fields, u-velocity contour, v-velocity contour, pressure contour are presented in Fig. 10-13.

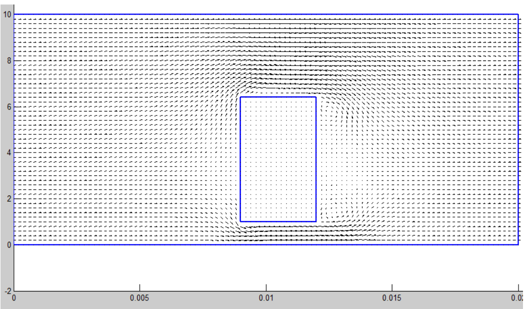

Figure 10. Velocity vector field (h=0.0002, nt=100)

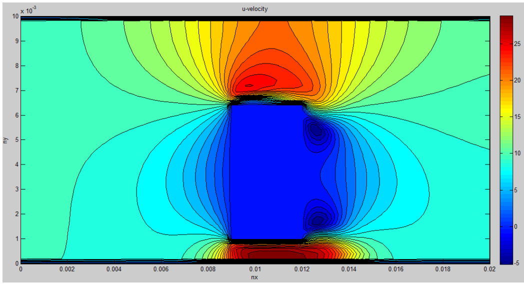

Figure 11. u-velocity contour (h=0.0002, nt=100)

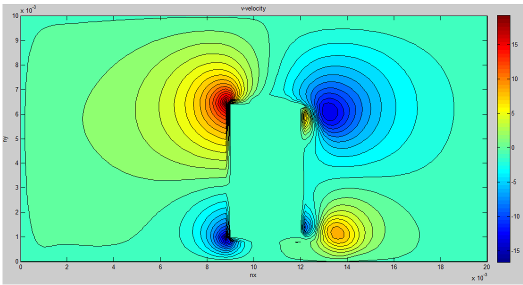

Figure 12. v-velocity contour (h=0.0002, nt=100)

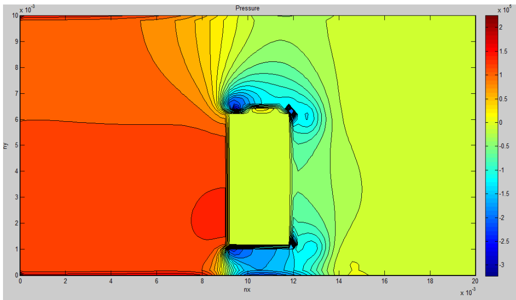

Figure 13. Pressure contour (h=0.0002, nt=100)

6. CONCLUSIONS

This paper presented an applicable method for simulating the wave body interaction problems. This method is cip (constrained interpolation profile). Throughout the research, we obtained some results as follows: from the physical phenomena, in particular here is the flow through an object in three phase environments (solid, liquid, gas). Then, we proceed to discretize these mathematical equations to create an algorithm and used computer to find the solution. This study uses matlab software as a tool for programming and presenting the results as graphs.

This paper has built a solver for two dimensional flows in a two phase (liquid, solid) environment. These results can be used to develop a three phase flow (liquid, air, and solid) [5].

Due to limited on the basis of information technology, mathematical knowledge, and fluid dynamic, this paper stops at the simulation of two phases flow problems and much remains unresolved, specifically error analysis and validation by experimental results.

In order to develop this work, it is necessary to analyze more simulations cases and invest more time. That is the future work. This method can be developed successfully to find the answer of three phase flow problem [6].

Acknowledgements: *This work was supported by the research grant of AUN/SEED-Net (JICA) over a total period of 2 years for Collaborative Research with Industry (CRI) project (Project No. HCMUT-CRI-1501).*

Bài toán tương tác giữa sóng nước và kết cấu sử dụng phương pháp CIP-Bài toán minh hoạ tính cho hai pha.

- **Trần Tiến Anh**
- **Bùi Quan Hùng**

Trường Đại học Bách Khoa, ĐHQG-HCM - ttienanh@yahoo.com

TÓM TẮT

Phương pháp CIP (Constrained Interpolation Profile) là một trong những phương pháp tính toán mô phỏng động lực học lưu chất (CFD) được phát triển bởi giáo sư người Nhật, Takashi Yabe. Nó được sử dụng để mô phỏng bài toán ba pha bao gồm không khí trên bề mặt, chất lỏng và kết cấu ở dạng rắn. Để kiểm tra tính chính xác của lý thuyết CIP, nhiều thí nghiệm với các bài toán khác nhau đã được thực hiện và thu được kết quả rất khả quan. Điều này chứng minh tính đúng đắn của phương pháp CIP. Căn cứ vào nhu cầu mô phỏng tương tác giữa sóng nước và kết cấu (sóng nước và phao của thủy phi cơ, thuyền bay, trụ bến tàu, giàn khoan, vỏ tàu ...), bài báo này áp dụng các lý thuyết của phương pháp CIP để tìm lời giải cho vấn đề của mô phỏng 2D của sóng nước qua một vật thể. Mục tiêu nghiên cứu là để hiểu biết rõ hơn về vật lý, tìm ra các phương trình vi phân mô tả hiện tượng này, sau đó tiến hành rời rạc hoá, thiết lập các thuật toán và tìm ra lời giải của phương trình. Bài viết này sử dụng phần mềm Matlab để viết các module chương trình và hiển thị kết quả.

Từ khóa: *giải thuật tính toán số, đường biên dạng nội suy, bài toán mặt thoáng, tương tác lưu chất và kết cấu, dòng nhiều pha, phương trình động lực học lưu chất.*

REFERENCES

[1]. Takashi Yabe, Feng Xiao, Takayuki Utsumi (2001). The constrained interpolation profile method for multiphase analysis. *Journal of Computational Physics 169*, pp. 556–593.

[2]. Kashiwaghi, M., Hu, C., Miyake, R. & Zhu. T. (2008). A CIP-based cartesian grid method for nonlinear wave-body interactions. *Nippon Kaiji Kyokai*.

[3]. Washino, K., Tan, H. S., Salman, A.D. & Hounslow, M.J. (2011). Direct numerical simulation of solid–liquid–gas three-phase flow: Fluid–solid interaction. *Powder Technology 206*, pp. 161–169.

[4]. Kishev, Z. R., Hu, C. & Kashiwagi, M. (2006). Numerical simulation of violent sloshing by a CIP-based method. *Journal of Marine Science and Technology*, Vol 11., pp. 111–122.

[5]. Shiraishi, K. & Matsuoka, T. (2008). Wave propagation simulation using the CIP method of characteristic equations. *Communications in Computational Physics*, Vol. 3, pp. 121-135.

[6]. Xiao, F. & Ikebata, A. (2003). An effcient method for capturing free boundaries in multi-fuid simulations. *International Journal for Numerical Methods in Fluids*, pp. 187–210.