FDM-FEM Coupled Method for Simulation of Interaction between Free Surface and Elastic Structure

Changhong HU[1], Kangping LIAO[2], Wenyang DUAN[3]

[1]*Research Institute for Applied Mechanics, Kyushu University, Kasuga, Fukuoka, JAPAN*

[2]*Interdisciplinary Graduate School of Engineering Science, Kyushu University, Kasuga, Fukuoka, JAPAN*

[3]*Deepwater Engineering Research Center, Harbin Engineering University, Harbin, CHINA*

ABSTRACT: In this paper a numerical method for interaction between free surface and an elastic structure with large deformation is proposed. A FEM (Finite Element Method) is used for solving dynamic structural deformation and a FDM (Finite Difference Method) method based on the CIP based Cartesian grid method (Hu et al, 2008), is applied for calculating the free surface flow. The FDM and FEM are coupled by a block Gauss-Seidel procedure. A validation computation result is presented on a tank sloshing problem with an elastic plate installed on the bottom.

1. INTRODUCTION

In rough sea conditions, marine structures such as ship, platform, etc. frequently experience wave impacts, which give extremely large impact forces on the structures and result in a shudder or elastic vibration of the structures. Investigation of such fluid-structure interaction (FSI) problem is of considerable importance in engineering application and has attracted much attention. The wave impact problem is different from other FSI problems such as an elastic body immersed in a fluid or a fluid bounded by elastic walls, due to its more complicated hydrodynamic features with free surface and its interaction with the structures. A successful calculation of such wave impact problem requires accurate modeling of the violent free surface variation and the free surface impact on the structure. Further, structure deformation may be an important factor and need to be considered in the computation. The objective of this study is to develop a numerical method based on coupling Finite Difference Method (FDM) with Finite Element Method (FEM) for investigation of the FSI problems involving free surface impact and structure deformation.

The concept of our approach is shown in Fig.1. The free surface flows are solved by the CIP based Cartesian grid method (Hu et al, 2004, 2009), the dynamic deformation of the elastic body is solved by a finite element method. Coupling of these two methods is carried out by the block Gauss-Seidel procedure (Joosten *et al*. 2009; Wood *et al*. 2008), which belongs to the partitioned approaches. Immersed Boundary (IB) method (Peskin, 1972) is used in this partitioned approach.

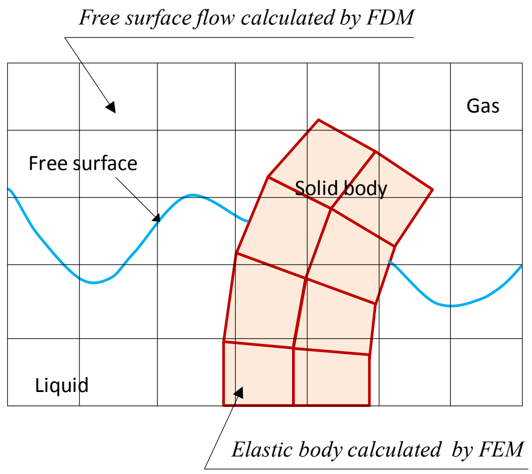

Fig.1 Coupled FDM and FEM method

In this paper, the numerical method is first described and a preliminary numerical result is presented with the comparison to available experimental date for validation.

2. MUMERICAL METHOD

The numerical method described in this paper is an extension of the CIP based Cartesian grid method, which has four key features: (1) a Cartesian grid approach for wave-body interactions; (2) the CIP (Constraint

Interpolation Profile, Yabe *et al.* 2001) method as the flow solver for water and air; (3) THINC (Tangent of Hyperbola for Interface Capturing; Xiao et al, 2005; Yokoi, 2007) scheme for interface capturing of the free surface; and (4) an immersed boundary method for fluid-structure interaction treatment.

2.1 Flow Solver

The governing equations of an unsteady, viscous and incompressible flow are as follows:

$$\nabla \cdot \mathbf{u} = 0 \tag{1}$$

$$\frac{D\mathbf{u}}{Dt} = -\frac{1}{\rho}\nabla p + \frac{1}{\rho}\nabla \cdot \mathbf{T} + \mathbf{f} \tag{2}$$

In which, **T** and **f** stands for the body force, such as gravity, etc. Time evaluation of Eq. (2) is performed by a fractional step method in which the equation is divided into an advection step and two non-advection steps. The advection calculation is performed by the CIP scheme. The pressure is treated in a non-advection step calculation, in which the following Poisson equation is solved.

$$\nabla \cdot \left(\frac{\nabla p}{\rho}\right) = \frac{1}{\Delta t}\nabla \cdot \mathbf{u} \tag{3}$$

Equation (3) is assumed valid for liquid, gas and solid phase. Solution of it gives the pressure distribution in the whole computation domain. The pressure distribution obtained inside the solid body is a fictitious one, which satisfies the divergence free condition of the velocity field. The pressure on the body boundaries is obtained by interpolation of the pressures in the FDM cells, and used as the external forces for structural analysis.

2.2 Structural Solver

The motion of structure can be described by:

$$[\mathbf{M}]\{\ddot{\mathbf{S}}\} + [\mathbf{C}]\{\dot{\mathbf{S}}\} + [\mathbf{K}]\{\mathbf{S}\} = \{\mathbf{F}(t)\} \tag{4}$$

Where $[\boldsymbol{M}]$, $[\boldsymbol{C}]$ and $[\boldsymbol{K}]$ are the Mass matrices, the damping matrices and the Stiffness matrices of structure, respectively. $\{\boldsymbol{F}(t)\}$ is the force vector acting on the structure, which depends on the time. $\{\boldsymbol{S}\}$, $\{\dot{\boldsymbol{S}}\}$ and $\{\ddot{\boldsymbol{S}}\}$ are the displacement, velocity and acceleration vectors respectively. Damping matrices are neglected in the present method.

Since the isoparametric elements make it possible to have elements with curved boundaries, it is adopted in our method to treat large deformation of the structure. The four nodes quadratic plane element is used to discrete the two-dimensional structure. Eq. (4) is solved by Newmark method.

2.3 Immersed Boundary Treatment for Flow simulation

For fluid-structure interaction problem, especially involving structure deformation, the interface between the fluid and structure should be treated carefully. In this paper, as the first step of the FSI research, we chose a relatively simple way, which is similar to Immersed Boundary Method (Peskin, 1972).

In the fluid solver, density function ϕ_m is used to define water, air and solid body, where $m = 1,2,3$ represents water, air and solid respectively. ϕ_1 is solved by THINC method, ϕ_3 is determined by the method that is described in (Hu and Kashiwagi, 2009), and $\phi_2 = 1 - \phi_2 - \phi_3$. After ϕ_m for all phases were determined, the physical properties, such as density, viscosity, etc. for each cell can be calculated by:

$$\lambda = \sum_{m=1}^{3} \phi_m \lambda_m \tag{5}$$

Then the velocity at the body boundary cell $\hat{\mathbf{U}}^{n+1}$, which is required to specify as the boundary condition for the flow solver, can be calculated by using the velocities at those particles. In the present numerical solution procedure, the method of imposing the velocity distribution inside and on the body boundary is equivalent to adding a forcing term to the momentum equation. The following updating operation is done after the calculation of Eq. (2).

$$\mathbf{u}^{n+1} = \phi_3 \hat{\mathbf{U}}^{n+1} + (1 - \phi_3)\mathbf{u}^{**} \tag{6}$$

In boundary cell Eq. (6) is a volume fraction weighting treatment for velocity interpolation.

2.4 Block Gauss-Seidel Procedure for Fluid Structure Interaction

In fluid structure interaction analysis, algorithmic treatment of the interface conditions between fluid and structure domain is one of the key points. In recent years, weakly and strongly coupled methods have been developed to solve this kind of problems. It should be noticed that definition of the weakly and strongly coupled is different from one to another. For convenience, we are using the definition that the direct method which uses one scheme for both fluid and structure is defined as monolithic approach. The method that uses different schemes for fluid and structure is defined as the partitioned approach, which can be further divided into weakly coupled and strongly coupled method. The main difference between the weakly and strongly coupling is the way of the information exchange at the interface. In a weakly coupled procedure, the information between two subsystems is exchanged only once per time step, while in a strongly coupled procedure, an iterative method is used to correct the initial values for variables until convergence is reached at each time step. On the other hand, the so-called block Gauss-Seidel procedure means that the subsystems (or blocks) are solved subsequently in every time step. The blocks are defined here as the fluid and structural domains. With the block Gauss-Seidel method, the fluid and structure domain can be solved separately by different schemes. The interaction between these two domains is achieved by communicating interface information as boundary conditions.

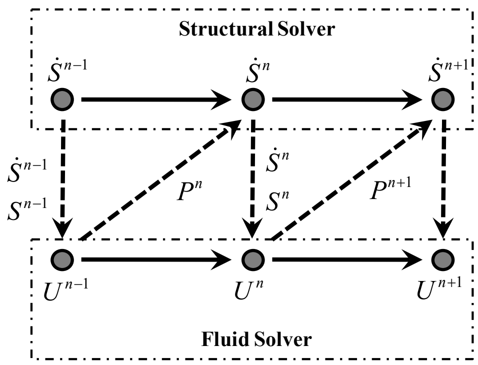

Fig. 2 block Gauss-Seidel procedure

The block Gauss-Seidel procedure used in the present study is shown in Fig.2. It is a strongly coupled method. From the structural solution results, displacement S^{n-1} and velocity $\dot{S}^{n-1}$ on the structure surface are used to give boundary conditions on the fluid domain. Then the flow simulation is performed to obtain the hydrodynamic force P^{n}. Subsequently, the hydrodynamic force is used as the external force acting on the structure surface, and structure dynamic equations are solved to get new displacement and velocity. These new displacement and velocity are then used for the next iterative step until convergence is achieved.

3. NUMERICAL RESULTS

As the first validation computation using the proposed numerical method, a tank sloshing problem with an elastic structure installed on the bottom is considered. The experiment data are taken from the paper (Idelsohn S.R *et al.* 2008). In the computation, a non-uniform Cartesian grid (206×132) is applied for fluid solver and the structure is discretized into 92 4-node elements. Variable time step, which is determined by the CFL condition, is used and the initial time interval is $\Delta t = 2 \times 10^{-4}$s.

Comparison between the present numerical result and the experiment on the time history of the horizontal displacement of the plate top is shown in Fig.3. Fairly good agreement is found on this comparison on the consideration that the tank motion for the computation could not be specified as the same as the experiment.

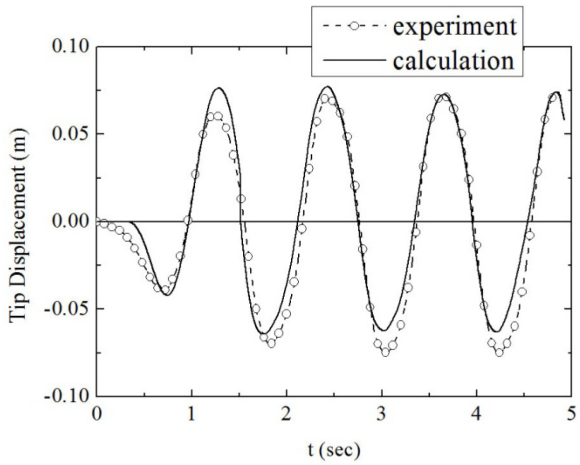

Fig.3 Time history of horizontal displacement at the top of the plate

The structure deformation and free surface profile at some typical time steps are shown in Fig.4. Comparing with the experimental results, the free surface profile of numerical simulation is not so smooth, and there are some bubbles in the fluid. The reason of these phenomena will be investigated in our future work.

4. CONCLUSIONS

In this paper, a coupling procedure with FDM and FEM is developed to investigate the interaction between free surface and an elastic structure involving large deformation. A two-dimensional numerical simulation on a tank sloshing case shows that the present method has great potential for such FSI problem. For the future works, improvement of the coupling procedure as well as the three dimensional code development will be carried out.

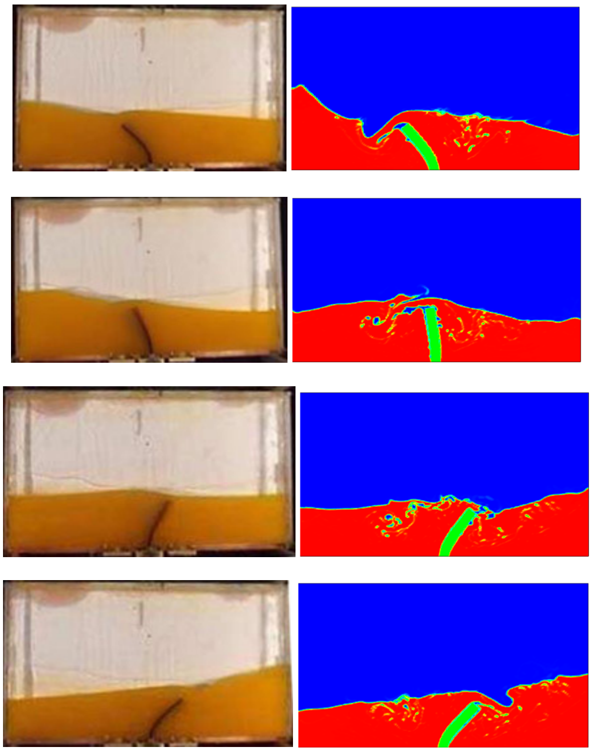

Fig. 4 Comparison between experimental and numerical (from top t=1.84, 2.12, 2.32, 2.56s)

REFERENCE

Hu, C.H. and Kashiwagi, M. (2004). "A CIP-based method for numerical simulations of violent free surface flows", *J Mar Sci Technol* . Vol. 9, pp143–157.

Hu, C.H. and Kashiwagi, M.(2009). "Two-dimensional numerical simulation and experiment strongly nonlinear wave–body interactions", *J Mar Sci Technol* . VOl.14, pp200–213.

Idelsohn, S.R. Marti., J., Souto-Iglesias, A. and Onate, E. "Interaction between an elastic structure and free-surface flows: experimental versus numerical comparious using the PFEM". *Comput Mech*. Vol. 43, 125-132. 2008

Joosten, M. M., Dettmer, W. G. and Peri, D (2009). "Analysis of the block Gauss–Seidel solution procedure for a strongly coupled model problem with reference to fluid–structure interaction". *Int. J. Numer. Meth. Engng*; Vol.78, pp757–778

Peskin, C. S. (1972), "Flow patterns around heart valves". *J. Comput. Phy*. Vol. 10, pp252-271

Wood, C et al. (2008), "Partitioned block-Gauss–Seidel coupling for dynamic fluid–structure interaction", *Comput Struct*, doi:10.1016/j.compstruc.2008.08.005

Xiao, F., Honma, Y. and Kono, T. (2005), "A simple algebraic interface capturing scheme using hyperbolic tangent function", *Int. J. Numer. Meth. Fluid*. Vol.48, pp1023.

Yabe, T, Xiao, F, and Utsumi, T. (2001), "The constrained interpolation profile method for multiphase analysis", *J Comput Phys.* Vol.169, pp556–593.

Yokoi, K (2007). "Efficient implementation of THINC scheme: A simple and practical smoothed VOF algorithm", *J. Compt.Phys*. Vol. 226, pp1985-2002.