# A New Approach to the Coupled Fluid-Structure Problem: An Example of a Circular Cylinder with an Elastic Surface in Uniform Flow

Hidehito IWASE ,Ryutaro HIMENO and Hideki MATSUMOTO

Computer Information Center and Advanced Engineering Center, RIKEN
2-1, Hirosawa, Wako-shi,Saitama 351-0198 Japan
E-mail, iwase@postman.riken.go.jp
Computer Information Center and Advanced Engineering Center, RIKEN
2-1, Hirosawa, Wako-shi,Saitama 351-0198 Japan
E-mail, himeno@postman.riken.go.jp
Science Technology System, NEC Information Systems
3-2-1, Sakado, Kozu-ku,Kawasaki,Kanagawa 213-0012 Japan
E-mail, matumoto@ssd.nis.nec.co.jp

**Abstract** : Recent applications of biomechanics have involved numerical schemes to solve a combined fluid-structure problem (CFS problem). It is particularly difficult to deal with a CFS problem which includes substantial geometric nonlinearity and so on. We report here a new scheme based on the loose coupling method for solving a CFS problem on biomechanics. The loose coupling method, which consists of the finite difference method for fluid analysis and the finite element method for elastic analysis, proceeds with the computation, and exchanging the data between fluid analysis and elastic analysis. To confirm the validity of this proposed new scheme, we solve the problem of a circular cylinder with an elastic surface in uniform flow as an example of a CFS problem.

## 1. INTRODUCTION

The purpose of this paper is to develop the new scheme for solving the CFS problem by biomechanics. In the studies on biomechanics simulation, it is necessary to treat large deformation in an elastic analysis because human tissue such as the brain, eye and skin are made of hyperelastic materials. The difficulies of an elastic analysis which involves the geometric nonlinearity and so on therefore have to be addressed. Here, we treat a circular cylinder with an elastic surface in uniform flow as an example of large deformation. Studies on a circular cylinder in uniform flow have been done so far for cases involving vibration or small deformation of steel material. In the case of large deformation in a short time, the absolute acceleration becomes very large. Consequently, the effect of this large acceleration has to be considered in the governing differential equation, so that a dynamic elastic analysis inevitably has to be adopted to solve the CFS problem. In studies of large deformation involving the CFS problem, the model of the elastic body has been replaced by a one-dimensional trass structure to simplfy the problem. We don’t adopt the model of trass structure, but instead adopt a two-dimensional elastic model.

Many methods have been developed for solving the CFS problem: CIP (cubic interpolation polynomial) method, ALE (Arbitrary Lagrangian Eulerian) method and loose coupling method are examples of the main ones. The loose coupling was selected for this present study because it has the advantage of ease in producing the required software. The analyses of the fluid dynamics and of the deformation of an elastic body were performed by finite difference method and finite element method, respectively. Ohba have experimented the flow around a elasto-flexible cylinder in uniform flow like our adopted model in this paper. Their results were referred to in our numerical results.

## 2. BASIC EQUATIONS OF THE CFS PROBLEM

### 2.1 BASIC EQUATIONS OF FLUID DYNAMICS

The flow around a circular cylinder is assumed to be incompressible.The governing equations of unsteady incompressible viscous flow are the equation of continuity(1) and Navier-Stokes equations (2)

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

$$\frac{\partial \mathbf{u}}{\partial t} + \{(\mathbf{u} - \mathbf{u_0}) \cdot \nabla\}\mathbf{u} = -\nabla p + \frac{1}{\mathrm{Re}}\nabla^2 \mathbf{u} \tag{2}$$

where $\mathbf{u}$ is the velocity vector of the fluid, $\mathbf{u}_0$ is the velocity vector of the grid system, $p$ is the static pressure and Re is the Reynolds number.

These equations are discretized by the finite-difference method on the boundary-fitted coordinate system. Treatment of the convective terms is very important for the stability in high-Reynolds-number flow. A third-order upwind scheme is chosen for discretization of the convective terms, and a second-order central scheme is used for the other spatial terms. For the time integration terms, a first-order Euler implicit scheme is employed instead of a higher-order time integration schemes. The MAC method is used to couple the velocity and the pressure fields.

## 2.2 BASIC EQUATIONS OF THE ELASTIC BODY

We shall consider here the two-dimensional elastic problem, assuming isotropic linear elastic model for the governing equation of the elastic body. The kinematic equation of the elastic body is as follows:

$$\sigma_{ij,j} + \bar{f}_i = \rho \ddot{u}_i \tag{3}$$

where $\sigma_{ij,j}$ , $\ddot{u}_i$ and $\rho$ respectively denote the stress, acceleration and density, and $\bar{f}_i$ is the volumetric force. The overscored point indicates derivation with respect of time, and the overscored bar is the prescribed data at the boundary.

The geometric boundary conditions are indicated as follows:

$$u_i = \bar{u}_i \qquad (\text{ on } S_u) \tag{4}$$

The dynamic boundary conditions are indicated as follows:

$$\sigma_{ij} n_j = \bar{p}_i \qquad (\text{ on } S_\sigma) \tag{5}$$

where $S_u$ and $S_\sigma$ denote the real displacement and stress boundaries.

Considering layleigh damping in the foregoing equations, we can apply the formulation of finite element method to the above equations. Accordingly, the kinematic equation is derived as follows:

$$\mathbf{M}\ddot{\mathbf{u}} + \mathbf{C}\dot{\mathbf{u}} + \mathbf{K}\mathbf{u} = \mathbf{F} \tag{6}$$

where $\dot{\mathbf{u}}$ is the velocity vector of the displacement, and $\mathbf{M}$, $\mathbf{C}$ and $\mathbf{K}$ are the mass, damping and stiffness matrices.

The damping matrix consists of viscous damping and structural damping, so that the dumping matrix $\mathbf{C}$ can be defined as follows:

$$\mathbf{C} = \alpha \mathbf{M} + \gamma \mathbf{K} \tag{7}$$

where $\alpha$ and $\gamma$ are user-specified parameters.

We can numerically analyze the foregoing kinematic equations by applying the Newmark $\beta$ method, which is known as one type of direct integral method. The equations used by Newmark $\beta$ method are shown as follows:

$$\ddot{\mathbf{u}}(t+\Delta t) = \left\{\mathbf{M} + \frac{\Delta t}{2}\mathbf{C} + \beta\Delta t^2\mathbf{K}\right\}^{-1} \begin{bmatrix} \mathbf{f}(t+\Delta t) - \mathbf{C}\left\{\dot{\mathbf{u}}(t) + \frac{\Delta t}{2}\ddot{\mathbf{u}}(t)\right\} \\ -\mathbf{K}\left\{\mathbf{u}(t) + \Delta t\dot{\mathbf{u}}(t) + (\frac{1}{2} - \beta)\Delta t^2\ddot{\mathbf{u}}(t)\right\} \end{bmatrix} \tag{8}$$

$$\dot{\mathbf{u}}(t+\Delta t) = \dot{\mathbf{u}}(t) + \frac{\Delta t}{2}\left\{\ddot{\mathbf{u}}(t) + \ddot{\mathbf{u}}(t+\Delta t)\right\} \tag{9}$$

$$\mathbf{u}(t+\Delta t) = \mathbf{u}(t) + \frac{\Delta t}{1!}\dot{\mathbf{u}}(t) + \frac{\Delta t^2}{2!}\ddot{\mathbf{u}}(t) + \beta\Delta t^3\,\frac{\ddot{\mathbf{u}}(t+\Delta t) - \ddot{\mathbf{u}}(t)}{\Delta t} \tag{10}$$

where $\beta$ is a parameter specified by the user. $\beta$ is given as 0.25 in this paper.

## 3. ALGORITHM OF COMPUTING

We will briefly review the new scheme for the CFS method proposed in this paper. Fig 1 shows the flow chart for the new scheme. Firstly, the mass matrix, damping matrix, stiffness matrix and coefficient matrix for the Newmark $\beta$ method are computed. Next, the iteration stage is input by time steps,and the pressure and the velocity for fluid dynamics analysis are computed by using the MAC method in conjunction with the finite difference method. The resulting pressures obtained by the fluid dynamics analysis become the loading conditions for the finite element method, and the displacement, velocity and acceleration of each node are computed by using the Newmark $\beta$ method in conjunction with finite element method. This computation procedure is repeated the required number of times. In this study, we apply the grid velocity to the Navier-Stokes equation. The equations for the velocity grid are as follows:

$$u_0 = \frac{x_i^{n+1} - x_i^n}{\Delta t} \tag{11}$$

$$v_0 = \frac{y_i^{n+1} - y_i^{n+1}}{\Delta t} \tag{12}$$

where $u_0$ is the x-directional grid velocity, and $v_0$ is the y-directional grid velocity.

## 4. COMPUTATION EXAMPLE

The two-dimensional circular cylinder consisting of a partial elastic surface is shown in Fig. 2. The diameter of the circular cylinder is 0.038m. The surface angled $\theta = 90^{\circ}$ on circular cylinder at upper side is rigid(internal of circular cylinder is rigid). Another surface consists of elastic material. The thickness of the elastic surface is 0.004m. The domain in the flow field is divided into a 182×182 grid by elliptical grid-generating method, the O-type grid topology is chosen for this grid system. Time steps of 0.00001 are used for both fluid dynamics and elastic analyses, and Reynolds number is given as 10000. We investigated the the deformation state of the elastic surface at each non-dimensional times $T = 20.0$(case 1), 60.0(case 2), 80.0(case 3), assuming that the surface of the circular cylinder was rigid until the specified non-dimensional time. The surface of the circular cylinder then suddenly changed to partial hyperelastic surface, whereby, the domain of the elastic body was divided into 272 triangular elements. The interpolation function for the displacement of each element is defined by a one-dimensional function.The parameters which determine the character of the damping matrix are given as $\alpha = 0.02$ and $\gamma = 0.05$. Poisson's ratio is taken as 0.4 and Young's modulus as 11,000Pa.

The boundary conditions are as follows:

Elastic surface domain

The boundary of the elastic surface of the circular cylinder is in the free condition.

The internal boundary of the circular cylinder is in the fixed condition.

The loading on the elastic surface is identified by the fluid dynamics pressure.

Fluid domain

The surface boundary of the circular cylinder is $u = 0$, $\frac{\partial p}{\partial n} = 0.0$

The external boundary of flow field is $u_x = 1.0$, $p = 0$

## 5. COMPUTED RESULTS

In case 1, a pair of vortices is generated behind the circular cylinder in the flow field. Fig. 4 shows the y-directional amplitude at point a, the impact force resulting from the fluid pressure vibrating the elastic surface of the circular cylinder. The traveling wave generated on elastic surface causes this amplitude. The mechanism for the generation of the traveling wave is initiated by the fluid impinging on the elastic surface. Consequently, the elastic surface is subjected to a large deformation. Furthermore, the impinging flow resulting in this large deformation causes a positive pressure field ahead of the large deformation and a negative pressure field behind it. The positive pressure field pushes the large deformation along the surface, while the negative pressure field pulls the large deformation along the surface. This consequent movement of the large deformation becomes the traveling wave as shown in Fig. 5. The deformation of the circular cylinder finally converges to the shape as shown in Fig. 6. In the case of a generated pair of vortices, the deformation becomes symmetrical since the distribution of pressure on both side is the same. These results conform with those of Ohba's experiments. The velocity vector is shown in Fig. 7, the flow pattern being almost the same as that without the elastic surface.

In case 2, a large vortex and the small vortex are generated at the upper right and lower right in Fig. 2 respectively. The deformation and the pressure contours are shown in Fig. 8. Since the circular cylinder is pulled toward the upper right place where the large vortex exists with its negative pressure, the deformation of the elastic surface on the shape indicated in Fig.8. We can observe the results that the traveling wave is continuously generated. Fig. 9 shows the y-directional amplitude at point a.

In case 3, a Karman vortex is induced. The pattern of the deformation is shown in Fig. 10, the elastic surface being pulled toward the Karman vortex.

## 6. CONCLUSIONS

We have proposed a new scheme based on the loose coupling method for solving the CFS problem. A circular cylinder with an elastic surface in uniform flow was taken as the example. Consequently, the deformation pattern and the mechanism for the generation of a traveling wave could be elucidated by our scheme. The results indicate that this method can be used to simulate the CFS problem.

## REFERENCES


[1] Yabe, T., and Aoki, T., A universal solver for hyperbolic equations by cubic-polynomial interpolation I. One-dimensional solver, Comp. Phy. Comm., **66**, 219-232,1991

[2] Shimura, M. and Sekine, A., Interaction analysis between structure and fluid flow for wind engineering, J. Wind Engrg. Indust. Aerodyn., **46&47**, 595-604, 1997.

[3] Ohnishi, R., Kimura, T., Miyatake, H., and Kaku, T., An Approach for Coupled Aero-Structure Analysis: Example of Multi-Disciplinary Application on Multiple Parallel Computers. Aerospace numerical simulation technology symposium, 47-52(in Japanese).

[4] Nakata, M., and Ohba, K., Drag and Flow Field around Elasto-Flexible Cylinder in Uniform Flow, Trans. Japan. Soc. Mech. Eng.(C), 1997, **63**, 674-679 (in Japanese).

[5] Tajikawa, T., Ohba, T., and Iwai, M., Transient Wave Traveling on the Surface of Elasto-Flexible Cylinder in Uniform Water Flow(Influence of Internal Fluid's Viscosity upon the Wave), 1999 Japan Computational Mechanics Convention, 295-276(in Japanese).