대한기계학회 2000년도 춘계학술대회논문집 B pp. 709~714 KSME 00S272

Splitting method for the combined formulation of fluid-particle problem

Hyunggwon Choi*, Jung Yul Yoo* and D. D. Joseph**

Key Words : Splitting method, Combined formulation, Fluid-particle problem, Pressure equation

Abstract

A splitting method for the direct numerical simulation of solid-liquid mixtures is presented, where a symmetric pressure equation is newly proposed. Through numerical experiment, it is found that the newly proposed splitting method works well with a matrix-free formulation for some bench mark problems avoiding an erroneous pressure field which appears when using the conventional pressure equation of a splitting method. When deriving a typical pressure equation of a splitting method, the motion of a solid particle has to be approximated by the 'intermediate velocity' instead of treating it as unknowns since it is necessary as a boundary condition. Therefore, the motion of a solid particle is treated in such an explicit way that a particle moves by the known form drag (pressure drag) that is calculated from the pressure equation in the previous step. From the numerical experiment, it was shown that this method gives an erroneous pressure field even for the very small time step size as a particle velocity increases. In this paper, coupling the unknowns of particle velocities in the pressure equation is proposed, where the resulting matrix is reduced to the symmetric one by applying the projector of the combined formulation. It has been tested over some bench mark problems and gives reasonable pressure fields.

1. Introduction

For the last two decades, the segregated finite element method has emerged as one of the useful tools to simulate large scale flow problems mainly because it is effective in memory use. However, with respect to the direct numerical simulation of fluid-particle mixture flows based on an unstructured mesh, there is no reported result from the segregated finite element method yet. As far as the author knows, only Glowinski et al. [1] adopted a theta scheme(a variation of the fractional step method) for the finite element direct numerical simulation of solid-liquid mixture flows. It is based on a structured background mesh and uses a Lagrangian multiplier to impose the constraint of a solid particle motion.

*School of mechanical and aerospace engineering, Seoul National University
*Aerospace Engineering and Mechanics, University of Minnesota

Hu [2] developed a direct numerical simulation finite element code based on an unstructured mesh for fluid-particle problems. Johnson and Tezduyar [3] also solved the same problems based on the different formuation. Both studies use an integrated formulation, where both velocity and pressure are obtained at the same time by solving the global system of the Navier-Stokes equations. Note that Johnson and Tezduyar use P1P1 element with a stabilized finite element method and Hu uses a conventional mixed P2P1 element with the combined formulation for the implicit treatment of a particle's motion. While Hu used an ALE formulation for the simulation of a moving grid (particle), Johnson and Tezduyar used DST/ST (Deforming-Spatial-Domain/Space-Time) procedure [4]. In the author's viewpoint, there are two additional important differences between the two studies. First, the method of Johnson and Tezduyar allows the use of an effective diagonal preconditioning due to the stabilized finite element formulation, but Hu's method has a difficulty in using a diagonal preconditioning [5] since the mixed finite element formulation of the Navier-Stokes equations generates the unsymmetric saddle point problem. Consequently, in the view point of a parallel computation Hu's formulation has some weak points compared to the Johnson and Tezduyar's approach. For more details of the problems related to the parallel computation of Hu's method, see the reference [5]. Secondly, Hu's method uses the combined formulation proposed by Hesla [6] for a 'coupled' fluid-particle system , where the force acting on a particle surface by a fluid need not be calcaulted explicitly since the force is implicitly linked with the particle velocity variables in terms of the fluid variables (velocity and pressure). On the other hand, in the method of Johnson and Tezduyar the coupling between the fluid variables and the particle variables is achieved by updating the particle position and velocity based on the fluid forces acting on the particle surface, which is calculated at each non-linear iteration within each time step.

In the present study, a splitting method for the combined mixed finite element formulation of fluid-particle problem is developed and tested with a matrix free approach. The main difference of the present method from existing splitting methods lies in deriving the SPD(symmetric positive definite) pressure equation. The new SPD pressure equation is proposed for the splitting method of the combined finite element formulation for fluid-particle problems and tested over the well known two dimensional benchmark problems. They are the single cylinder sedimentation in a channel, 2 particle sedimentation in a closed box with *drafting, kissing and tumbling* [7] and 100 particle sedimentation in a closed box. The newly proposed pressure equation for the combined formulation of fluid-particle problem gives satisfactory results for the benchmark problems with a diagonal preconditioning thus enabling us to use a matrix-free approach.

In Section 2, a summary of the splitting method for the combined formulation of fluid-particle problem is given and in Section 3, a pressure equation for the combined formulation is proposed and discussed. Finally, in Section 4, typical benchmark problems are tested to verify the developed splitting code.

2. Numerical Method

There are many kinds of split methods (projection method or fractional step method) depending on time marching methods (explicit , implicit or semi-implicit), the accuracy of the temporal discretization, or how to get a pressure equation from the divergence free constraint. In the present study, a four step fractional step method [8] is used with P2P1 mixed finite element and the second order accurate fully implicit Crank-Nicolson time marching scheme is used. In the first step of the present fractional step method, the Navier-Stokes equations coupled with the Newton's law of a solid particle motion are solved, where the pressure term is decoupled from those of convection, diffusion and other external forces. At this step, the intermediate velocity $\hat{u}$ is obtained, which does not necessarily satisfy the continuity equation. Hence, at the following step, the pressure is to be obtained from the continuity constraint and the velocity is corrected by that pressure field.

In the first step, the following set of equations are to be solved.

$$\begin{aligned} &\frac{\hat{u}_i - u_i^n}{\Delta t} + \frac{1}{2}\tilde{u}_j^n \hat{u}_{i,j} + \frac{1}{2}\tilde{u}_j^n u_{i,j}^n = \hat{\sigma}_{ij,j} + \sigma_{ij,j}^n + S_i^n \\ &M\frac{\hat{U}_p - U_p^n}{\Delta t} = G_p + F_p \end{aligned} \qquad (1)$$

$U_p = \begin{bmatrix} u_p \\ v_p \\ \omega_p \end{bmatrix}$, $\quad M = \begin{bmatrix} m & 0 & 0 \\ 0 & m & 0 \\ 0 & 0 & I \end{bmatrix}$ for the two dimensional case

, where $\hat{u}_i$ is the 'intermediate velocity', u_i^n is the fluid velocity in the previous time step, $\tilde{u}_i^n = u_i^n - u_{i,g}^n$ is the convection velocity in ALE formulation with a grid velocity $u_{i,g}^n$, S_i = external forces , G_p is external force acting on the particle, F_p is the force acting on the particle surface by the fluid that is not necessary to be calculated due to the combined formulation used in the present study, m is the mass of the particle and I is the moment of inertia of the particle. Since the fluid is Newtonian, $\hat{\sigma}_{ij} = \nu D_{ij}(\hat{u})/2$ and $\sigma_{ij}^n = -p^n \tilde{\mathrm{I}} + \nu D_{ij}(u^n)/2$, where p is the pressure, ν is the kinematic viscosity (constant in the present study), and $D_{ij}(\hat{u})$ is the rate of strain tensor. Note that the linearized convection term is used in the present study to avoid the nonlinear iteration. In the present study, the combined formulation by Hesla [6] is used for the fluid-particle mixture simulation. Therefore, unlike conventional split methods, all velocity components are calculated in a coupled manner. The Galerkin formulation of the above equation can be written as follows:

Find $u_i^h(\mathbf{x},t) \in H_h^1(\Omega), U \in R \times R$ and $\omega \in R$ such that

$$\int_\Omega [w_i^h(\frac{\hat{u}_i^h - u_i^n}{\Delta t} + 0.5*\tilde{u}_j^n \hat{u}_{i,j}^h + 0.5*\tilde{u}_j^n u_{i,j}^n) + \nabla w_i^h : (\hat{\sigma}_{ij}^h + \sigma_{ij}^n)] d\Omega - \int_\Omega w_i^h (\hat{\sigma}_{ij}^h + \sigma_{ij}^n) n_j d\Gamma + \sum_p \delta U_p (M \frac{dU_p}{dt} - G_p) = \int_\Omega w_i^h S_i^h d\Omega \quad (2)$$

for all admissible weight function $w_i^h \in V_h$ with the following boundary conditions

$$\begin{aligned} \boldsymbol{u} &= \mathbf{g} && \text{on } \Gamma_g \\ \sigma \cdot \mathbf{n} &= \boldsymbol{h} && \text{on } \Gamma_h \\ \boldsymbol{u} &= \boldsymbol{U}_p + \omega_p \times (\mathbf{x} - \mathbf{X}_p) && \text{on } \Gamma_p \end{aligned}$$

where, $\mathbf{n}$ is the normal vector to the boundary Γ_h and the $\mathbf{x}$ is the coordinate of a node on the particle surface Γ_p and $\mathbf{X}_p$ is the coordinate of the center of the particle and

$$\mathbf{V}_h = \{ w_h | w_h \in H_h^1(\Omega), w_h = 0 \text{ on } \Gamma_g, w_h = \delta \mathbf{U}_p + \delta\omega_p \times (\mathbf{x} - \mathbf{X}_p) \text{ on } \Gamma_p \text{ for p} = 1,...., \mathrm{N}_p \}$$

For more details of the weak formulation of the combined approach, see references [2,6]. Therefore, in the first step the resulting matrix is written as follows:

$$\begin{bmatrix} A & E \\ C & D \end{bmatrix} \quad (3)$$

D: diagonal matrix

E: sparse matrix from the kinematic constraint

C: matrix from combined formulation.

Note that the block A becomes a symmetric matrix for the semi-implicit method, where the convection term is treated in a explicit way. In the second step, the following set of equations are to be solved.

$$\frac{u_i^* - \hat{u}_i}{\Delta t} = + \frac{1}{\rho} p_{,i}^n \quad (4)$$

$$M \frac{U_p^* - \hat{U}_p}{\Delta t} = F_p$$

SPD pressure equation for fluid-particle mixture problem

In the modified splitting code, the velocities of particles are solved with fluid variables in a coupled way so that the motion of the particles can be linked with the pressure equation implicitly. The governing equation is the coupled equation for u_i, U_p, p at time step n+1.

$$\frac{u_i^{n+1}-u_i^*}{\Delta t}=-\frac{1}{\rho}p_{,i}^{n+1} \qquad (8)$$

$$M\frac{U_p^{n+1}-U_p^*}{dt}=F_p$$

$$u_{i,i}^{n+1}=0$$

Applying a weak formulation for the combined formulation of fluid-particle problem, the resulting matrix is

$$\begin{bmatrix} M & -B & E \\ B' & 0 & 0 \\ C_M & C_P & D \end{bmatrix}\begin{bmatrix} u_i \\ p \\ U \end{bmatrix}=\begin{bmatrix} f_u \\ 0 \\ f_U \end{bmatrix} \qquad (9)$$

where M is mass matrix and B is gradient matrix. Note that the resulting matrix is an unsymmetric one due to the combined formulation. On the other hand, a symmetric matrix corresponding to Eq. (9) can be obtained using the projector proposed by Maury and Glowinsky [9]. In their approaches, the fluid velocity variables are divided into internal variables and variables on the particle surface, $u=[u_I,u_\Gamma]$. Then, the resulting matrix for the combined formulation is written as follows:

$$\begin{bmatrix} \tilde{M} & -\tilde{B} \\ \tilde{B}' & 0 \end{bmatrix}\begin{bmatrix} \tilde{u} \\ p \end{bmatrix}=\begin{bmatrix} \tilde{f}_u \\ 0 \end{bmatrix}$$

where, $\tilde{M}=\begin{bmatrix} D+P'M_\Gamma P & P'M_{\Gamma I} \\ M_{I\Gamma}P & M_I \end{bmatrix}$ $\quad \tilde{B}=\begin{bmatrix} P'B_\Gamma \\ B_I \end{bmatrix}$

with $u_\Gamma = PU$ and $\tilde{u}=[U\ u_I]^T$

3. Numerical Results

In order to verify the proposed algorithm, single cylinder sedimentation in a two dimensional channel, the sedimentation of two cylinders in a closed box and the sedimentation of 100 cylinders in a closed box are selected as the two dimensional benchmark problems. In the single cylinder sedimentation case, the density of the particle is 1.01 , the density of fluid is 1.0, kinematic viscosity is 0.01, the channel width is 20.0 and the particle diameter is 0.5 in CGS units. Fig. 1 (a) shows the configuration of the particle and the finite element mesh at the selected time step with the boundary conditions used. The number of pressure unknowns is about 3,300, the number of velocity nodes on the particle surface is 100, and the Reynolds number based on the diameter, the average terminal velocity of the particle and the kinematic viscosity of the fluid is about 122. Note that in the present paper, only vertex nodes (nodes for pressure variables in P2P1 element) are shown in all unstructured mesh figures, so the mesh resolution for the velocity variables of an unstructured mesh is about 4 times larger than that shown in the corresponding figure. Since the ALE formulation is adopted in the present study, the entire computational domain is moving to the streamwise direction (gravity direction) with the streamwise direction velocity of the particle at each time step. Fig. 1 (b)~(d) show the isobars, streamlines and isovorticity lines at the selected time. In Fig. 1 (d), the dotted lines represent the negative isovorticity lines. Fig. 2 shows the time histories of translation and angular velocities of the sedimenting single cylinder. From the Fig. 2, the period of the oscillation of the sedimenting cylinder is 1.343 sec (0.745 Hz) and the corresponding Strouhal number is 0.152. Note that in the case of the periodic vortex shedding around the fixed cylinder, the corresponding Strouhal number is 0.175 from Swanson [10].

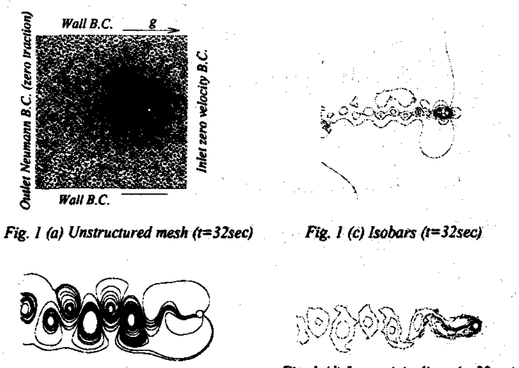

Fig. 1 (a) Unstructured mesh (t=32sec) *Fig. 1 (c) Isobars (t=32sec)*

Fig. 1 (b) Streamlines (t=32sec) *Fig. 1 (d) Isovorticity lines (t=32sec)*

Fig. 1 Single cylinder sedimentation in a channel

The second simulation is done to see the well known *drafting, kissing and tumbling* scenario [7]. In this case, the density of the particle is 1.02 , the density of fluid is 1.0, kinematic viscosity is 0.01, the box size is 4×20 and particle diameter is 0.5 in CGS units. Fig. 3 shows the configurations of two cylinders and the corresponding finite element meshes at various time steps. From Fig. 3, we can see the well known *drafting, kissing and tumbling* phenomenon. Fig. 4 shows the two cylinders' position and isobars at the selected time steps. Fig. 5 shows isovorticity lines at the selected time steps. In this simulation, the number of pressure unknowns is about 5,500, and the number of velocity nodes on the particle surface is 100 at minimum. The maximum Reynolds number based on the diameter, the translation velocity of the particle and the kinematic viscosity of the fluid is about 270 during the entire simulation.

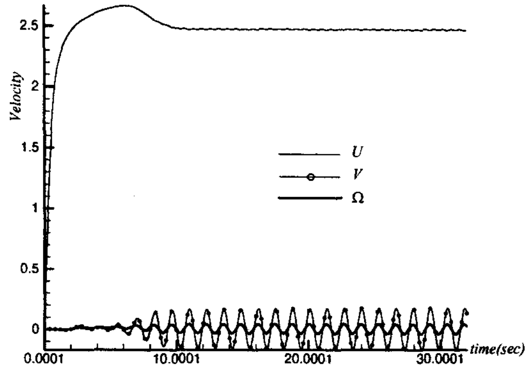

Fig. 2 The time histories of translation and angular velocities of the sedimenting cylinder

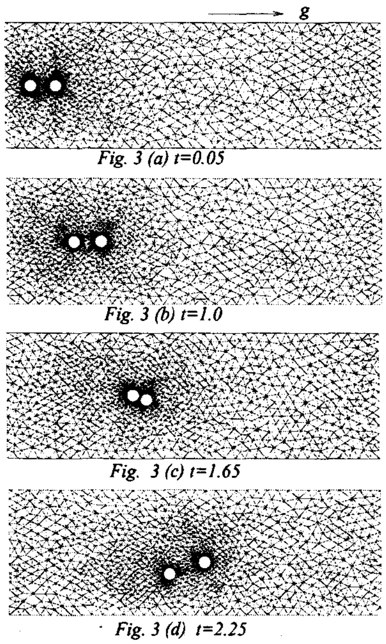

Fig. 3 Unstructured meshes at various time steps

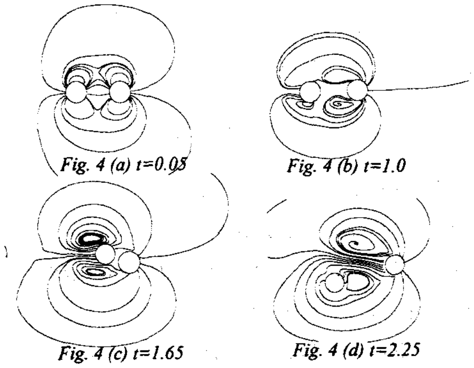

Fig. 4 Streamlines at various time steps

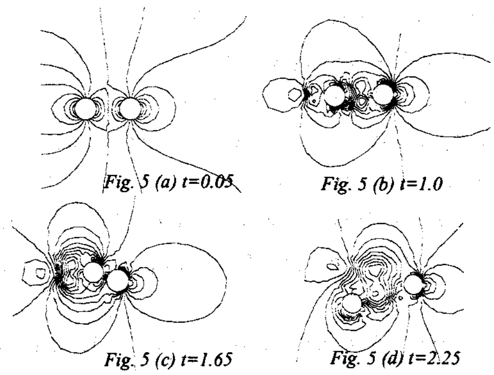

Fig. 5 Isovars at various time steps

4. Conclusion

In this paper, a fractional step method for the combined formulation of the fluid-particle mixture problems is presented. It has been shown that a new SPD (symmetric positive definite) pressure equation should be used in order to get reasonable pressure fields in the present study. From the test of the benchmark problems, the proposed splitting method gives reasonable results reproducing the well known *drafting, kissing and* *tumbling* scenario. Various preconditioners for the SPD pressure equation are to be studied.

Acknowledgement

This work is supported by NSF under KDI/NCC grant NSF/CTS-9873236, the Minnesota Supercomputing Institure (MSI) and Turbo and Power Machinery Research Center (TPMRC).

References

[1] R. Glowinski, T. -W. Pan, J. Periaux , Distributed Lagrange multiplier methods for incompressible viscous flow around moving rigid bodies, Comp. Meth. Appl. Mech. Engrg. 151 (1998) 181-194.

[2] H.H. Hu, Direct simulation of flows of solid-liquid mixtures, Int. J. Multiphase Flow. 22 (1996) 335-352.

[3] A.A. Johnson and T.E. Tezduyar, Simulation of multiple spheres falling in a liquid-filled tube, Comput. Meth. Appl. Mech. Engrg. 134 (1996) 351-373.

[4] T.E. Tezduyar, J. Liou and M. Behr, A new strategy for the finite element computations involving moving boundaries and interface - the DSD/ST procedure : I. The concept and the preliminary numerical tests. Comput. Meth. Appl. Mech. Engrg. 94 (1992) 339-351.

[5] H.G. Choi, Parallel computation of the combined finite element formulation for particle-fluid mixture problem, Unpublished note, 1998.

[6] T.I. Hesla, Combined formulation of fluid-particle problem, Unpublished note, 1991.

[7] A.F. Fortes, D.D Joseph and T.S. Lundgren, Nonlinear mechanics of fluidization of beds of spherical particles, J. Fluid Mech. 177, 467-483.

[8] H.G. Choi, H.Choi and J.Y.Yoo, A fractional four-step finite element formulation of the unsteady incompressible Navier-Stokes equations using SUPG and linear equal-order element methods, , Comput. Methods Appl. Mech. Engrg. 143 (1997) 333-348

[9] B. Maury and R. Glowinski, Fluid-particle flow : A symmetric formulation, C. R. Acad. Sci. Paris. 324 (1997) 1079-1084.

[10] Swanson, W.M., Fluid Mechanics, Holt, Rinehart and Winston (New York,1970) Section 11.5