Journal of Applied Mechanics Vol.9, pp.151-159 (August 2006) JSCE

Solid-fluid interaction analysis by using a multi-material Eulerian finite element method

Rafael Doig, Shigenobu Okazawa and Masahiko Fujikubo

Department of Social and Environmental Engineering, Hiroshima University
(1-4-1 Kagamiyama, Higashi-Hiroshima 739-8527)

A multi-material Eulerian finite element method for simulating solid-fluid interaction analyses is proposed in this paper. Although a conventional solid-fluid interaction code models solid and fluid in a Lagrangian and an Eulerian formulation respectively, the present computational framework bases on just the Eulerian formulation for solid and fluid. A mixture theory is used for handling multi-material (solid and fluid) elements. Solid and fluid element stresses are then mixed by considering their density function.

Key Words : *Solid-fluid interaction, Eulerian FEM; multi-material; mixture theory*

1. Introduction

A computation of solid-fluid interaction problems are nowadays no longer un-researched [1),2)]. The most famous solid-fluid interaction has a Lagrangian solid and an Eulerian fluid model, which usually implies two solvers. Both solvers interact during the simulation. Initially, fluid pressure is given as load to the solid. After the solid solver has carried out the calculation, resulting solid velocities are given to the fluid as load. Then, the fluid solver calculates. This loop remains until a desired time or until solution converges.

The objective of this paper is to present a new approach for solid-fluid interaction analysis. All materials are modeled in same domain, an Eulerian Domain. Mixed elements are then handled by a mixture theory. We expect to take the advantages of the Eulerian formulation even for solids, as like large deformations and material separation. In this publication only the quality of solid-fluid interaction is presented. Quantitative results will be conducted later.

It is to be pointed out that present computational framework bases on an existing explicit finite element program for solid dynamic problems [3)]. The implementation of the fluid solving capacity considers that. For incompressible viscous fluid, the Navier-Stokes equations which represent the equation of momentum conservation have commonly four terms: inertial, internal and external forces. Differentiating those equations from them of solid dynamics, only the stress is calculated in another way. In other words, different constitutive equations are used. Additionally velocities are updated due to the fluid pressure effect.

2. Computational Framework

The computational framework is introduced in this section. The following three sections describe the three main parts of the solver: the solid, fluid and solid-fluid interaction part, respectively. Before starting with details, it is to be pointed out that the solution of this fixed-mesh finite element method is splitted here in two steps, a Lagrangian step and an Eulerian or advective step. Later step transports the material and corresponding solution variables through the remapped mesh. Both steps, the Lagrangian and Eulerian are solved at each time step. The flowchart of a computational step is described in below box. After reading and initializing the program data, a loop is started until the wished end time is reached.

Loop: Flowchart of a computational step

- **Lagrangian step**
  - + calculate element stress
    - - solid stress
    - - fluid stress
    - - average stress
  - + calculate internal nodal forces
  - + calculate nodal accelerations
  - + calculate nodal velocities
    - - calculate fluid pressure
    - - update nodal velocities
  - + calculate coordinates
- **Eulerian or advective step**
  - + advect solution variables
  - + advect nodal velocities

At first, in the **Lagrangian step**, the element stress is calculated. Here, stress laws for both materials are required, Hooke's law for the solid and Newtonian fluid law. Notice that only shear stresses due to fluid viscosity are calculated. The key of this study is the mixture of the element stress. Considering the density function of each material, solid and fluid, the stress is averaged. A mixture theory handles a multi-material element, which implies that more than one material in the element are in contact or in other words are interacting, here solid and fluid. As next, internal nodal forces are calculated by considering previous averaged element stress. Then, the nodal accelerations and velocities are computed. As mentioned before, only fluid shear stresses were considered. From now on, the fluid pressure is calculated. The well-known fractional step method has been implemented. Thus, the pressure Poisson equation is solved globally by using previous calculated velocities. A later step updates the velocities considering the pressure effect. Nodal coordinates are then calculated, which means the mesh has moved.

The Eulerian or advective step re-meshs to the original configuration. As consequence, material and corresponding solution variables must be transported. For the two kind of variables, namely, element and nodal-centered, two different algorithms are used. A second-order advection algorithm advects element-centered variables, like stress, density function, etc. Only velocities are nodal-centered variables. To satisfy the conservation of momentum, nodal momentum is advected instead of the velocity only. The nodal-centered advection algorithm does this job. The Eulerian step is solved based on the finite difference method rather than the finite element method applied in the Lagrangian step.

It is worth to mentioned that implemented subroutines integrate the element with a single gauss point, which simplify the interaction of solution variables between solid and fluid.

3. Solid Dynamics

Theory details about implemented finite element method solid dynamics can be found in the literature [4),5),6),7)]. Reviewing the Eulerian formulation of a continua problem, let's assume an arbitrary solution variable ϕ. The relation between the material and the spatial time derivatives is

$$\frac{D\phi}{Dt} = \frac{\partial \phi}{\partial t} + \boldsymbol{v} \cdot (\nabla \phi) \tag{1}$$

where $D\phi/Dt$ is the material time derivative and $\partial\phi/\partial t$ the spatial time derivative of the solution variable ϕ. $\boldsymbol{v}$ is the velocity field. The difference makes the convective term $\boldsymbol{v} \cdot (\nabla\phi)$. It can be shown that the Eulerian governing equations, namely the mass, momentum and energy conservation equations follow the general conservative form

$$\frac{\partial \phi}{\partial t} + \nabla \cdot \boldsymbol{\Phi} = S \tag{2}$$

where $\boldsymbol{\Phi}$ is a flux function and S is a source.

The operator split method splits Eq. (2) into two equations, namely a Lagrangian step, Eq. (3) and an advective step, Eq. (4). These equations are solved sequentially. The solution of Eq. (3), which is advanced in time, is used. In Eq. (4) the first term of left-hand side is a dummy value, since it's solved by keeping the time stopped.

$$\frac{\partial \phi}{\partial t} = S \tag{3}$$

$$\frac{\partial \phi}{\partial t} + \nabla \cdot \boldsymbol{\Phi} = 0 \tag{4}$$

Equation (3) is the so-called Lagrangian step. So, the mass, momentum and energy conservation are solved in this step

$$\frac{\partial \rho}{\partial t} = 0 \tag{5}$$

$$\frac{\partial \rho \boldsymbol{v}}{\partial t} = \nabla \cdot \boldsymbol{\sigma} + \rho \boldsymbol{b} \tag{6}$$

$$\frac{\partial e}{\partial t} = \boldsymbol{\sigma} : \boldsymbol{D} \tag{7}$$

where ρ is the density of the material, $\boldsymbol{\sigma}$ is the Cauchy stress, $\boldsymbol{b}$ is the body acceleration, e is the internal energy and $\boldsymbol{D}$ is the velocity strain. These equations are solved in standard Lagrange hydrocodes, in explicit as well in implicit time integration. For impact problems the explicit time integration is more attractive because of its suitability to parallel computing. No global stiffness matrix is necessary to solved, making it easier to code.

4. Fluid Dynamics

4.1 Navier-Stokes Equations

The finite element method is widely and successfully applied also in fluid dynamics problems [8),9),10)]. Despite the advantages of the finite element in computational fluid dynamics, this study extends an existing finite element hydrocode for solid dynamics. The governing equations are splitted in a Lagrangian step followed by an Eulerian or advective step. The first step is done by using the finite element method while the Eulerian step is solved by the finite difference method.

The primitives variables of an incompressible viscous fluid dynamic process are mainly given by following the Navier-Stokes equations, namely, the momentum equation and the incompressibility constrain equation.

$$\rho\dot{\boldsymbol{v}} + \nabla p = \mu\nabla^2\boldsymbol{v} + \rho\boldsymbol{b} \tag{8}$$

$$\nabla \cdot \boldsymbol{v} = 0 \tag{9}$$

where ρ is the fluid density, $\boldsymbol{v}$ is the velocity field, p is the pressure, μ is the dynamic viscosity and $\boldsymbol{b}$ is the body acceleration. In a 2D problem with $\boldsymbol{v}^T = [u\ v]$, Eq. (8) contains two equations

$$\rho\frac{\partial u}{\partial t} + \frac{\partial p}{\partial x} = \mu\left[\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right] + \rho b_x \tag{10}$$

$$\rho\frac{\partial v}{\partial t} + \frac{\partial p}{\partial y} = \mu\left[\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right] + \rho b_y \tag{11}$$

4.2 Solving the Navier-Stokes Equations

A way of decoupling velocity and pressure is the concept of a fractional step (or projection or pressure-correction or Chorin's) method. Although accuracy might be lost, this method is widely applied due to the relative ease of implementation and computational performance.

A fractional step method splits the momentum equation in

$$\rho\dot{\boldsymbol{v}} = \nu\nabla^2\boldsymbol{v} + \rho\boldsymbol{b} \tag{12}$$

$$\rho\dot{\boldsymbol{v}} = -\nabla p \tag{13}$$

The concept of the fractional method is carried out in three steps:

1. the calculation of an intermediate velocity (neglecting pressure effects)

$$\boldsymbol{v}_*^{n+1} = \boldsymbol{v}^n + \frac{\Delta t}{\rho}\left[\nu\nabla^2\boldsymbol{v}^n + \rho\boldsymbol{b}\right] \tag{14}$$

2. the calculation of the pressure by considering the previously calculated intermediate velocity
   Taking the divergence or curl $(\nabla\cdot)$ of Eq. (13) and considering the incompressibility constrain of Eq. (9), the pressure Poisson equation is reached

$$\nabla \cdot \boldsymbol{v}^{n+1} = \nabla \cdot \boldsymbol{v}_*^{n+1} - \frac{\Delta t}{\rho}\nabla \cdot \nabla p \tag{15}$$

$$\nabla^2 p = \frac{\rho}{\Delta t}\left[\nabla \cdot \boldsymbol{v}_*^{n+1}\right] \tag{16}$$

3. the correction of the velocity by considering pressure effects

$$\boldsymbol{v}^{n+1} = \boldsymbol{v}_*^{n+1} - \frac{\Delta t}{\rho}\nabla p \tag{17}$$

The first step of the fractional step method is solved by using the explicit time integration. Different from the implemented solver for solid analysis, the stress in now calculated regarding the constitutive relation of a Newtonian fluid. The fluid shear stress vector is then calculated by

$$\boldsymbol{\tau} = \int_\Omega \mathbb{C}\mathbf{B}\boldsymbol{v}d\Omega \tag{18}$$

where the B is the strain-displacement matrix. $\mathbb{C}$ is the constitutive tensor as

$$\mathbb{C} = \mu\begin{bmatrix} 1 & -1 & 0 \\ -1 & 1 & 0 \\ 0 & 0 & 1 \end{bmatrix} \tag{19}$$

4.3 Discrete Poisson Equation

The pressure Poisson equation however is solved globally.

$$\left[C^T M_L^{-1} C + S\right] p^{n+1} = \frac{1}{\Delta t} C^T \boldsymbol{v}_*^{n+1} \tag{20}$$

where M_L is the lamped mass matrix. $C = \nabla$ is the gradient operator and $C^T = \nabla\cdot$ is divergence operator. S is the pressure stabilization matrix recommended by [11),12)]. Equation (20) yields to an equation system of the type **Ax=b** and can reach huge dimensions, therefore an iterative solver is more appropriate rather than a direct solver. Iterative solvers however are beyond of this content. As experienced here, it is just pointed out that an incomplete Cholesky conjugate gradient iterative solver works well.

5. Solid-Fluid Interaction

Mixture theories concept the treatment of more than one material in an element, i.e. two materials as shown in Fig. 1. The simplest mixture theory have the assumption that the strain rate $\dot{\boldsymbol{\epsilon}}$ is same for all present materials in an element. So the stress rate $\dot{\boldsymbol{\sigma}}_m$ of material m is given by

$$\dot{\boldsymbol{\sigma}}_m = \mathbb{C}_m : \dot{\boldsymbol{\epsilon}} \tag{21}$$

where $\mathbb{C}_m$ is the constitutive tensor of material m.

The updated stress $\boldsymbol{\sigma}_m^{n+1}$ by time $n+1$ is then

$$\boldsymbol{\sigma}_m^{n+1} = \boldsymbol{\sigma}_m^n + \dot{\boldsymbol{\sigma}}_m \Delta t \tag{22}$$

The element mean stress $\overline{\boldsymbol{\sigma}}^{n+1}$ is then

$$\overline{\boldsymbol{\sigma}}^{n+1} = \sum_{m=1}^{nmat} \boldsymbol{\sigma}_m^{n+1}\phi_m \tag{23}$$

where $nmat$ is the number of materials present in the element and ϕ_m is the element density function of each material m.

The simplest mixture theory, also called the mean strain rate mixture theory, is favored because of its simplicity and robustness [13),14)]. Here, we used it for a solid-fluid interaction problem.

Slide Condition: By neglecting the shear components of stress, we can simulate the slide condition.

$$\overline{P}^{n+1} = \sum_{m=1}^{M} P_m^{n+1}\phi_m \tag{24}$$

where P is the material pressure.

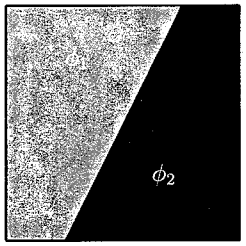

Fig. 1 Element containing 2 materials

6. Eulerian Advective Step

Equation (4) is often called the advective, convective or Eulerian step. This represents the material transport through the cells of the fixed Euler mesh. In the operator splitting method Eqs. (3) and (4) are solved sequentially as illustrated in Fig. 2. In order to extend existing traditional Lagrange finite element programs to an Eulerian formulation in a convenient way, practical-to-implement finite difference subroutines have been focused here. In this hydrocode, a monotonic, second order, cell-centered algorithm has been programmed for cell-centered solution variables, while a modification has been done for the momentum advection, which is vertex-centered. In this case, the vertex-centered momentum will be averaged to a cell-centered momentum, which is advected with the same algorithm for cell-centered variables. A final step is required, namely to extrapolate the cell-centered results to a vertex-centered ones [15),16),17)].

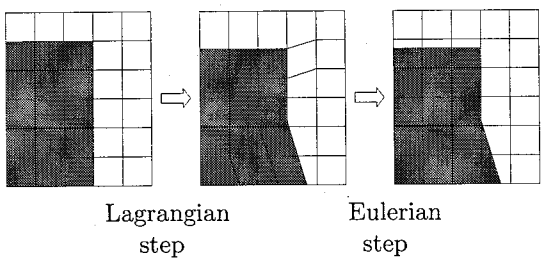

Fig. 2 Eulerian step after Lagrangian step

After the Lagrangian step, the deformed mesh is remeshed to the original place easily by setting nodal coordinates back. Here, it's important to remember that the deformed mesh is not allowed to have longer deformations than the element size. The Courant time step control method must take care of this. Here the second order Van Leer's MUSCL [17)] (monotone upwind schemes for conservation laws) method is implemented. MUSCL method is a one-dimensional transport algorithm. In logically meshes the one-dimensional advection is carried out along the mesh lines. Notice that advecting materials implies an automatically movement of interface between materials.

7. Computational Results

Calculations carried out here assumes an deformable body and an incompressible fluid. At first instance, present results try to show the quality of the solid-fluid interaction rather than giving a quantitative validation which is planned as next. The pressure-velocity exchange process while interacting can be clearly appreciated on present computation results. Material data, model geometry and simulation information as well results are presented. In all calculations the time step size is 0.00005 seconds. Since material used here are almost incompressible, the mass is conserved. Further, a finer mesh has been selected for reducing interface diffusion.

7.1 Ring Falling onto Fluid

In the first computation, an elastic ring falls free onto a fluid. A solid-fluid interaction simulation is target by expecting elastic ring deformation due to fluid pressure at contact surface as well fluid crater formations due to the ring buoyancy. Material properties are shown in Table 1. As for the geometry of the

Table 1 Material data: ring-fluid

ring	
Density [kg/m^3]	500
Young modulus [MPa]	0.05
Poisson's ratio [-]	0.30
fluid	
Density [kg/m^3]	1000
Dyn. viscosity [N.s/m^2]	0.001

model as shown in Fig. 3, the model size is 1.5×1.0m. The ring has an outer diameter of 0.25m and an inner diameter of 0.15. The height of the fluid is 0.4m. For simulating a certain impact velocity when the free-falling ring reaches the fluid, an initial velocity of 0.2m/s down-wards has been added to the elastic ring. The element division is 100×66. While the lower part of the ring is supported by the fluid at impact, the upper part bears a downward forcing due to gravity, 9.81m/s.. The result of this force combination forms an oval shape. Thanks the elasticity of the ring it tries to recover its initial shape as shown in the simulation plots from 0.0 to 0.9 seconds in Fig. 6. Xiao and Yabe [18)] published a similar simulation.

7.2 Bar in Fluid Flow

An at-the-bottom fixed elastic bar is loaded by a fluid flow, inflow from the left side and outflow at the right side as shown in Fig. 4. At calculation start

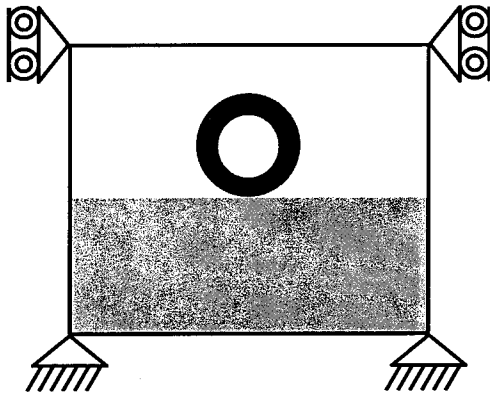

Fig. 3 Ring Falling onto Fluid

the velocity is zero and it will increase from the inflow boundary. Inflow and outflow are computational so handled that the volume fraction keep always 1. When the fluid velocity reaches the bar, an bar bending is expected due to fluid pressure at left side of the bar. Material properties are shown in Table 2. As

Table 2 Material data: bar-fluid

bar	
Density [kg/m^3]	1000
Young modulus [MPa]	5
Poisson's ratio [-]	0.30
fluid	
Density [kg/m^3]	1000
Dyn. viscosity [N.s/m^2]	0.001

for the geometry of the model as shown in Fig. 4, the model size is 1.0×1.0m. The bar has a size of 0.1×0.8m. The element division is 50×50. No gravitational acceleration is added. The flow velocity is 0.2m/s from left to right. Simulation plots from 0.0 to 2.4 seconds are shown in Fig. 7.

7.3 Pontoon Falling onto Fluid

A pontoon-wise structure impacts a fluid surface with an initial velocity simulating a certain height of free fall. Results show here also a well simulated solid-fluid interaction. At first the pontoon tries to displace the fluid which intends to scape at the sides. However, the fluid pressure at pontoon bottom at certain point tries to lift pontoon. The fluid at the sides occupy again the it initial place. The pontoon faces bending-swingings throughout all simulation. Material properties are shown in Table 3. As for the geometry of the model as shown in Fig. 5, the model size is 1.5×1.0m. The pontoon's bottom has a length of 1.0m and a thickness of 0.08m. The side-walls have a

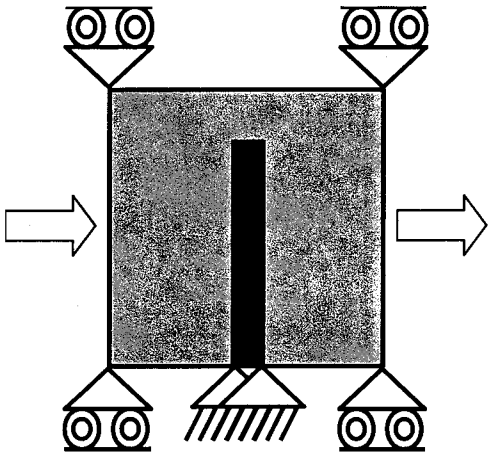

Fig. 4 Bar in Fluid Flow

Table 3 Material data: pontoon-fluid

pontoon	
Density [kg/m^3]	1500
Young modulus [MPa]	2
Poisson's ratio [-]	0.30
fluid	
Density [kg/m^3]	1000
Dyn. viscosity [N.s/m^2]	0.001

height of 0.4m and a thickness of 0.1m. The height of the fluid is 0.5m. For simulating a certain impact velocity when the free-falling ring reaches the fluid, an initial velocity of 0.5m/s down-wards has been added to the pontoon. The element division is 50×34. While

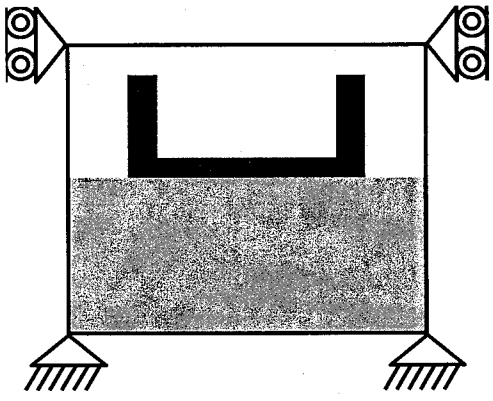

Fig. 5 Pontoon Falling onto Fluid

the lower part of the pontoon is supported by the fluid at impact, the weight (acceleration also 9.81m/s) of both side-walls bends the bottom. The simulation plots from 0.0 to 1.8 seconds are shown in Fig. 8.

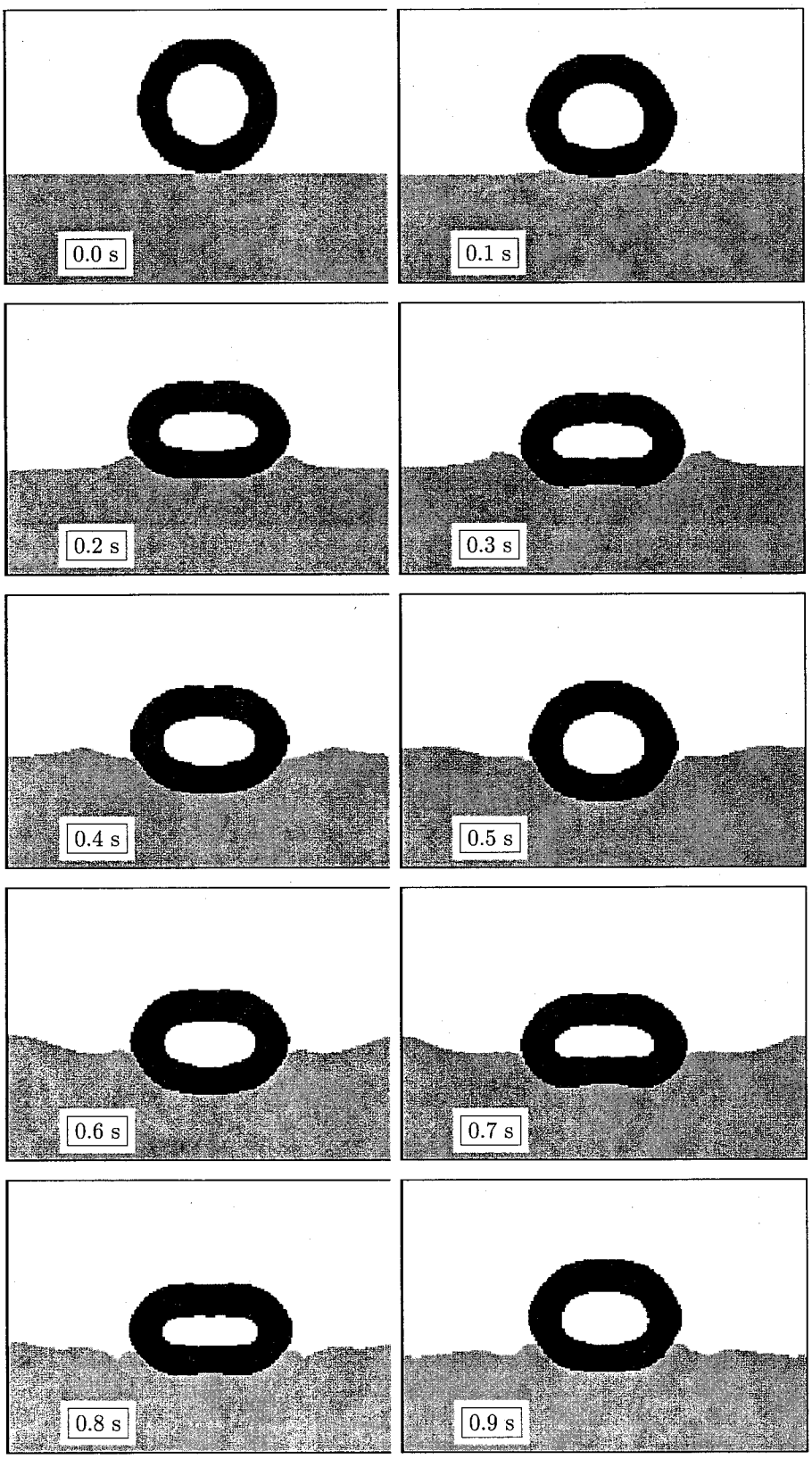

Fig. 6 Ring falling onto fluid: Eulerian deformation analysis

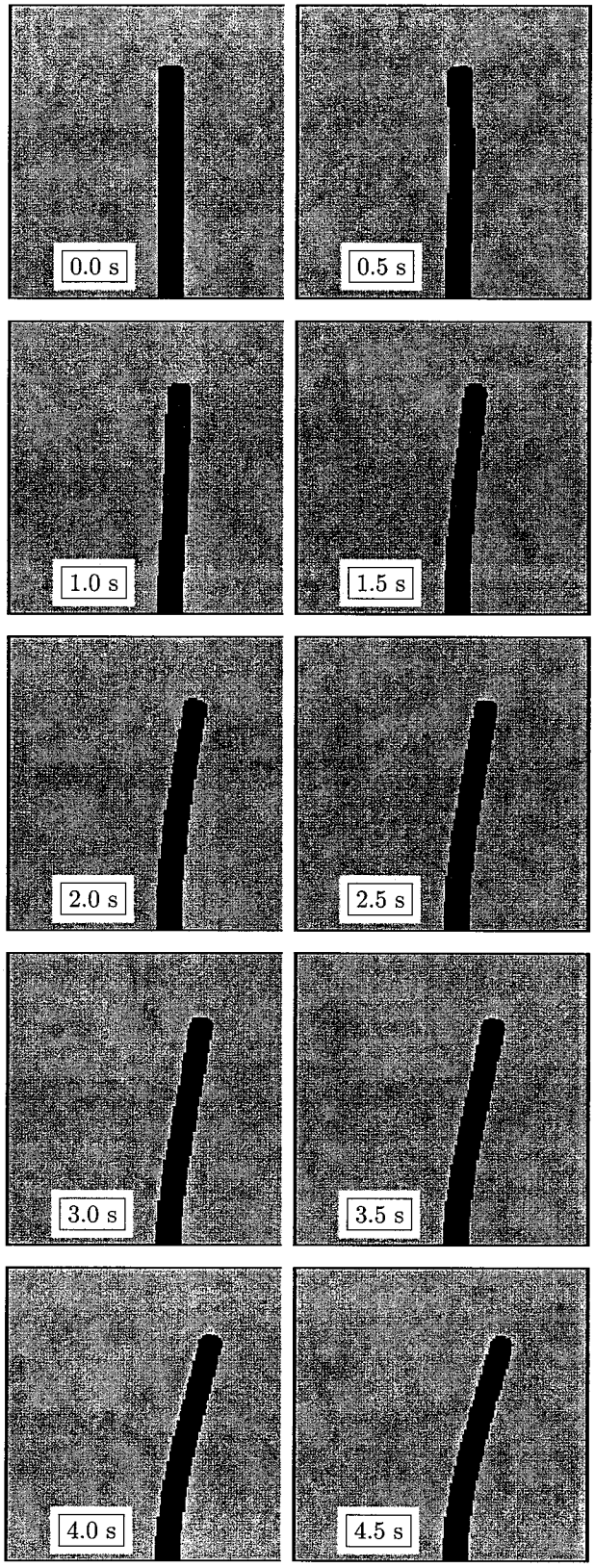

Fig. 7 Bar in fluid flow: Eulerian deformation analysis

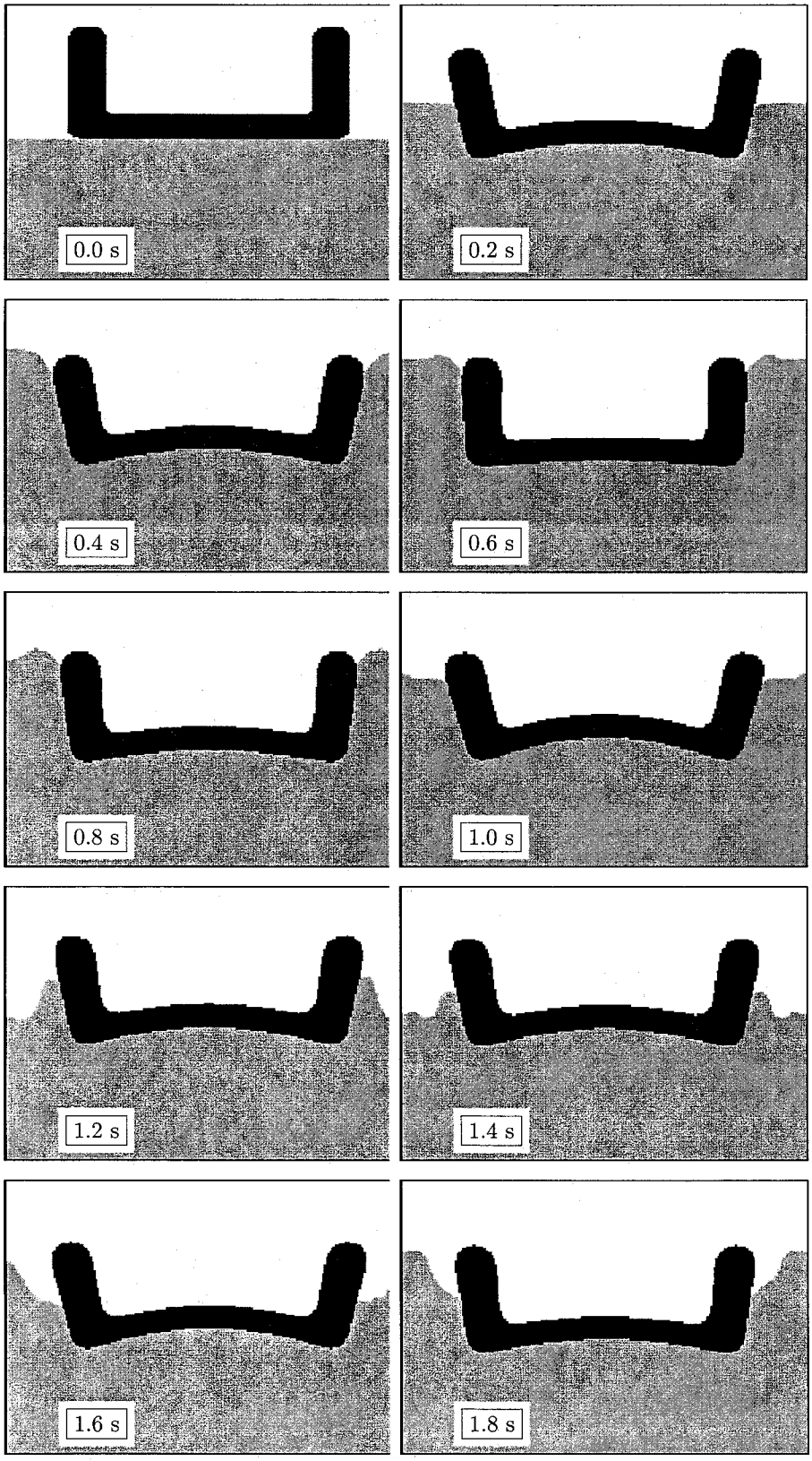

Fig. 8 Pontoon falling onto fluid: Eulerian deformation analysis

8. Conclusion

An Eulerian finite element method for simulating solid-fluid interaction problems has been presented here. Solid-fluid interaction tests with flexible or elastic bodies have been selected rather than rigid bodies. Interaction is well simulated as shown in the ring-fluid, bar-fluid and pontoon-fluid calculations. Buoyancy effects are also well simulated. In fact, calculations show promising results regarding the use of simple mixture theory. A complete stress mixture is carried out at each element rather than an interaction algorithm is used. Worth to mention is the fact that the computational framework has been adapted to the existing explicit finite element method without remarkable changes in code. This could encourage researchers on extending their existing Eulerian codes for further research on solid-fluid interaction with a mixture theory.

Regarding accuracy, a much accurate interaction process is reached with finer meshes, however the computational cost can increase rapidly. It's also not be hidden that an Eulerian solid faces a meshing limitation by regular-mesh solvers. Interesting also could be simulations considering more of the advantages of an Eulerian finite element approach. Problems like large deformation with mesh distortion, contact, creation of new free-surfaces after material separation are almost well overcome by using an Eulerian finite element method.

Acknowledgment

This work was financially supported by a scholarship of the Japanese Ministry of Education, Science and Culture to whom we would like to express our gratitude.

REFERENCES

1) Bathe K., Zhang H. and Ji S.: Finite element analysis of fluid flows coupled with structural interactions. *Computers & Structures*, Vol. 72, pp.1-16, 1999.
2) Le Tallec P. and Mouro J.: Fluid structure interaction with large structural displacements. *Computer Methods in Applied Mechanics and Engineering*, Vol. 190, pp. 3039-3068, 2001.
3) Doig R., Okazawa S. and Fujikubo M.: High-speed contact-impact simulations with Lagrangian and Eulerian hydrocode. *Journal of Applied Mechanics*, Vol. 8, pp.267-276, 2005.
4) Belytschko T., Liu W.K. and Moran B.: *Nonlinear finite elements for continua and structures*, John Wiley 2000
5) Holzapfel G.A.: *Nonlinear solid mechanics*, John Wiley 2001
6) Bonet J. and Wood R.D.: *Nonlinear continuum mechanics for finite element analysis*, Cambridge University Press 1997
7) Hallquist J.O.: *LS-DYNA theoretical manual*, Livermore Software Technology Corporation, May 1998
8) Gresho P.M., Sani R.L.: *"Incompressible flow and the finite element method"*, Vol.1 + Vol.2, John Wiley & Sons 1998
9) Gunzburger M.D: *Finite element methods for viscous incompressible flows*, Academic Press 1989
10) Chung T.J.: *Finite element analysis in fluid dynamics*, MCGraw-Hill 1978
11) Christon M.A.: Dealing with pressure: FEM solution strategies for the pressure in the time-dependent Navier-Stokes equations. *International Journal for Numerical Methods in Fluids*, Vol. 38, pp.1177-1198, 2002
12) Silverster D.J. and Kechkar N.: Stabilized bilinear-constant velocity-pressure finite elements for the conjugate gradient solution of the stokes problem. *Computer Methods in Applied Mechanics and Engineering*, Vol. 79, pp.71-86, 1990
13) Benson D.J.: An implicit multi-material Eulerian formulation. *International Journal for Numerical Methods in Engineering*, Vol. 48, pp.475-499, 2000.
14) Benson D.J. and Okazawa S.: Contact in multi-material Eulerian finite element formulation. *Computer Methods in Applied Mechanics and Engineering*, Vol. 193, pp.4277-4298, 2004.
15) Benson D.J.: Momentum advection on a staggered mesh. *Journal of Computational Physics*, Vol. 100, pp.143-162, 1992.
16) Benson D.J.: Computational methods in Lagrangian and Eulerian Hydrocodes. *Computer Methods in Applied Mechanics and Engineering*, Vol. 99, pp.235-394, 1992.
17) Van Leer B.: Towards the ultimative conservative difference scheme. IV. A new approach to numerical convection. *Journal of Computational Physics*, Vol. 23, pp.276-299, 1977.
18) Xiao F. and Yabe T.: Computation of complex flows containing rheological bodies. *Computational Fluid Dynamics Journal*, Vol. 8, pp.043-049, 1999

(Received April 13, 2006)