An Analysis of Two-phase Fluids Using Finite Element Method

Eri HATAUCHI

Department of Civil Engineering, Chuo University,
1-13-27, Kasuga, Bunkyo-ku, Tokyo 112-8551, Japan
E-mail: d37228@educ.kc.chuo-u.ac.jp

Abstract
The purpose of this study is to analyze two phase fluid flows of different density. As governing equations, the Navier-Stokes equation that considers surface tension and advection-diffusion equation to calculate the concentration is used. Compressibility is considered by using acoustic velocity for the Navier-Stokes equation. The two equations are calculated alternately. The calculation proceeds to the next time step if the new solution is calculated, which is equal to the previous solution. The Galerkin method is applied to the spatial discretization and the Crank-Nicolson method is applied to the temporal discretization. The mixed interpolation is used as interpolation technique based on the bubble function and linear function. In the normal Galerkin method, the advective term included in the equation of motion is unstable if the advection is dominated. This is solved by using the stabilization bubble function in this study. Two phase fluids are successfully analyzed using the present method.

KEY WORDS: finite element method, moving boundary problem, surface tension, Navier-Stokes equation, advection-diffusion equation, bubble function, acoustic velocity

1 INTRODUCTION

Recently, a simulation with computers is useful instead of experiments that need large-scale equipment, considerable cost and consuming time. Simulations with computers are used in various fields, and a lot of important results can be achieved.

The finite element method is frequently used in the engineering field. This is one of the most useful numerical analysis techniques. It is a method to obtain an approximate solution of the differential equation. In this method, the computational domain is divided into small areas, and interpolate using a certain function. The finite element method is developed in the field of structural analysis, but it is widely used also in other fields. It is a flexible method that can deal with various problems.

Moving boundary problem is a phenomenon that two objects move with contact face; liquid and gas, liquid and liquid, solid and gas, etc. Many flow phenomena have the moving boundary. It is necessary to consider the movement of the interface to analyze such fluids.

In this study, two phase fluids with different density with moving boundary is analyzed. An analysis of two phase fluids is useful to solve various engineering problems. For example, to analyze the behavior and its influence at the outflow accident of crude oil, or, influence on air when water is transported with a pipe etc.

The Navier-Stokes equation and advection-diffusion equation are used as governing equations. It is difficult to solve the simultaneous equations on the Navier-Stokes and advection-diffusion equation. Therefore, these equations are solved separately. It is calculated alternately to improve the accuracy of solution.
Acoustic velocity is used to solve the Navier-Stokes equation. It is known that the acoustic velocity is nearly constant in a nearly non-compressible fluid. Using acoustic velocity, it is possible to analyze the fluids considering density change. By considering compressibility, an analysis similar to an actual phenomenon is possible.

2 GOVERNING EQUATION

The Navier-Stokes equation and advection-diffusion equation are used as the governing equations. In this paper, indicial notation with summation convention is used.

2.1 ACOUSTIC VELOCITY

Using acoustic velocity, it is possible to analyze fluids considering density change. It is assumed that flows are independent on the temperature, and pressure $p$ is a function only of the density $\rho$.

$$p = p(\rho)$$ (1)

Equation for conservation of mass is as follows:

$$\dot{\rho} + u_j p_{,j} + \rho u_{i,i} = 0$$ (2)

where $u_i$ is velocity. Equation (1) is differentiated with respect to time $t$.

$$\frac{Dp}{Dt} = \frac{\partial p}{\partial \rho} \frac{D\rho}{Dt}$$ (3)

Equation (3) is expressed by (2) as follows;

$$\dot{\rho} + u_j p_{,j} + \rho c^2 u_{i,i} = 0$$ (4)

where $c$ is acoustic velocity.

$$c^2 = \frac{\partial p}{\partial \rho}$$ (5)

Equation (4) is the equation of continuity considering the effect of acoustic velocity. Equation of motion is expressed as follows:

$$\rho(\dot{u}_i + u_j u_{i,j}) + p_{,i} - (\lambda d_{kk,i} + 2\mu d_{ij,j}) = 0$$ (6)

$\lambda$ and $\mu$ are volume viscosity coefficient and shear viscosity coefficient, respectively, and $d_{ij}$ is expressed as follows:

$$d_{ij} = \frac{1}{2}(u_{i,j} + u_{j,i})$$ (7)

equation (6) is written by (7) as follows:

$$\rho(\dot{u}_i + u_j u_{i,j}) + p_{,i} - \lambda u_{kk,i} - \mu(u_{i,j} + u_{j,i}), j = 0$$ (8)

Equation (4) and (8) are used for the governing equations.
2.2 MOVING BOUNDARY PROBLEM

The Navier-Stokes equations (9)(10) and advection-diffusion equation (11) are employed as the governing equations. Dimensionless expression of equations (4) and (8) are (9) and (10).

\[
\begin{align*}
\rho \dot{u}_i + \rho u_j u_{i,j} + cp_{,i} - \lambda u_{k,k,i} - \mu (u_{i,j} + u_{j,i})_{,j} + k\phi_{,i} - \rho f_i = 0 & \quad \text{in } \Omega, \quad \text{(9)} \\
\dot{p} + u_j p_{,j} + cu_{i,i} = 0 & \quad \text{in } \Omega, \quad \text{(10)} \\
\dot{\phi} + u_j \phi_{,j} - \gamma \phi_{,ii} = 0 & \quad \text{in } \Omega, \quad \text{(11)}
\end{align*}
\]

where \( \Omega \) is computational domain, and \( f, k \) and \( \gamma \) are surface tension, concentration and diffusion coefficient and \( \phi \) is concentration. The area where \( \phi \) is \( \hat{\phi} \) can be decided as boundary \( \Gamma \). \( \Omega_{\phi>\hat{\phi}} \) is \( \Omega_1 \) and \( \Omega_{\phi<\hat{\phi}} \) is \( \Omega_2 \). \( \Omega_1 \) and \( \Omega_2 \) are the areas of fluid 1 and fluid 2 respectively. \( \rho \) is constant in each fluid.

Boundary condition are given in equations (12)-(14).

\[
\begin{align*}
u_1 & = 0, t_2 = 0 \quad \text{on } \Gamma_1, \quad \text{(12)} \\
u_2 & = 0, t_1 = 0 \quad \text{on } \Gamma_2, \quad \text{(13)} \\
t_i & = \{-cp\delta_{ij} + u_{k,k}\delta_{ij} + \nu(u_{i,j} + u_{j,i})\}n_j, \quad \text{(14)}
\end{align*}
\]

where \( \delta_{ij} \) is Kronecker delta.

\[\Gamma_1\]
\[\Gamma_2\]
\[\Omega_1\]
\[\Omega_2\]
\[\Gamma\]
\[\hat{\phi}\]
\[\phi\]
\[\phi=0\]
\[\phi>0\]
\[\phi<0\]

Figure 1 : Numerical model

2.3 SURFACE TENSION

A surface tension \( f \) is shown as follows;

\[
\hat{f}_i = \sigma \kappa n_i \quad \text{(15)}
\]

where \( \sigma, \kappa \) and \( n \) are surface tension coefficient, curvature, and unit normal vector from \( \Omega_1 \) to \( \Omega_2 \) on the boundary of two fluids, \( n_i \) is given.

\[
n_i = \frac{e_i}{|e|_{\phi=\hat{\phi}}} \quad \text{(16)}
\]

where \( e \) is gradient of concentration \( \phi \).

\[
e_i = \phi_{,i} \quad \text{(17)}
\]

Curvature \( \kappa \) is written as follows.

\[
\kappa = \frac{1}{|e|} \left[ \left( \frac{e_i}{|e|} \right) |e|_{,i} - e_{i,i} \right] \quad \text{(18)}
\]
Surface tension \( f \) is obtained by equation (9). It is introduced into the Navier-Stokes equation as the external force.

\[ \text{3 DISCRETIZATION} \]

The Galerkin method is applied to the spatial discretization and the Crank-Nicolson method is applied to the temporal discretization.

\[ \text{3.1 SPATIAL DISCRETIZATION} \]

The Galerkin method is applied to the spatial discretization. The weighting functions are shown by \( \omega_i, q, r \) and integrates over the computational domain. The weighted residual equation is expressed as follows;

\[
\rho \int_\Omega \omega_i \dot{u}_i d\Omega + \rho \int_\Omega \omega_i u_j u_i,j d\Omega + c \int_\Omega \omega_i p,i d\Omega - \lambda \int_\Omega \omega_i u_k,ki d\Omega - \mu \int_\Omega \omega_i u_i,jj d\Omega - \mu \int_\Omega \omega_i \phi,i d\Omega - \rho f_i = 0, \quad (19)
\]

\[
\int_\Omega q \dot{p} d\Omega + \int_\Omega qu u_i,j p,j d\Omega + c \int_\Omega qu u_i,i d\Omega = 0, \quad (20)
\]

\[
\int_\Omega r \dot{\phi} d\Omega + \int_\Omega ru \phi,i d\Omega - \gamma \int_\Omega r \phi,i d\Omega = 0, \quad (21)
\]

The mixed interpolation technique is applied. The linear element is used for the pressure. The bubble function element is used for the velocity and the concentration.

**Linear element (Figure 2)**

\[
p = \Psi_1 p_1 + \Psi_2 p_2 + \Psi_3 p_3, \quad (22)
\]

\[
\Psi_1 = L_1, \quad \Psi_2 = L_2, \quad \Psi_3 = L_3, \quad (23)
\]

**Bubble function element (Figure 3)**

\[
u_i = \Phi_1 u_{i1} + \Phi_2 u_{i2} + \Phi_3 u_{i3} + \Phi_4 u_{i4}, \quad (24)
\]

\[
u_{i4} = u_{i4} - \frac{1}{3} (u_{i1} + u_{i2} + u_{i3}), \quad (25)
\]

\[
\Phi_1 = L_1, \quad \Phi_2 = L_2, \quad \Phi_3 = L_3, \quad \Phi_4 = 27 L_1 L_2 L_3, \quad (26)
\]

Figure 2: Linear element  

Figure 3: Bubble function element

Numerical stability of the bubble function element is not satisfactory. The stabilization parameter is introduced at the center of the gravity.

\[
\tau_{eB} = \frac{<\phi_e,1>_A^2}{\Omega_e (\nu + \nu') ||\phi_e,j||_{\Omega_e}^2 A_e}, \quad (27)
\]

Controlled stabilizing parameter \( \tau_{eB} \) is set to be equivalent to \( \tau_{eS} \), which is used for the stabilization of streamline upwind finite element method.
\[ \tau_{eB} = \tau_{eS}, \]  
\[ \tau_{eS} = \left[ \left( \frac{2|u_i|}{h_e} \right)^2 + \left( \frac{4\nu}{h_e^2} \right)^2 \right]^{-\frac{1}{2}}, \]  
\[ <u,v> = \int_{\Omega_e} uv d\Omega, \]  
\[ ||u||_{\Omega_e}^2 = \int_{\Omega_e} uv d\Omega, \]  
\[ <\phi_c,1>_{\Omega_e} = \frac{A_c}{6}, \]  
\[ ||\phi_c||_{\Omega_e}^2 = 2A_c g, \]  
\[ g = \sum_{i=1}^{2} |\psi_{a,i}|^2, \]  
\[ he = \sqrt{2A_c}, \]

The spatial discretization of the state equation is shown as follows:

\[ \rho M_{\alpha\beta} \dot{u}_{\beta i} + \rho K_{\alpha\beta\psi j} u_{\beta j} u_{\psi i} - cH_{\alpha\psi} p_{\psi} + S_{\alpha\beta j} u_{\beta j} - kB_{\alpha i} \phi_{\psi} - \rho f_i = 0, \]  
\[ M_{\alpha\beta} p_{\psi} + K_{\alpha\beta\psi j} u_{\beta j} p_{\psi} + cH_{\alpha\psi} u_{\psi i} = 0, \]  
\[ M_{\alpha\beta} \phi_{\beta} - u_i A_{\alpha i \beta} \phi_{\beta} + \gamma D_{\alpha i \beta} \phi_{\beta} = 0, \]

\[ M_{\alpha\beta} = \int_{\Omega} (\Phi_{\alpha} \Phi_{\beta}) d\Omega, \]  
\[ K_{\alpha\beta\psi j} = \int_{\Omega} (\Phi_{\alpha} \Phi_{\beta} \Phi_{\psi j}) d\Omega, \]  
\[ H_{\alpha\psi} = \int_{\Omega} (\Phi_{\alpha i} \Phi_{\psi}) d\Omega, \]  
\[ S_{\alpha i \beta j} = (\lambda + \mu) \int_{\Omega} (\Phi_{\alpha i} \Phi_{\beta j}) d\Omega + \mu \int_{\Omega} (\Phi_{\alpha k} \Phi_{\beta k}) \delta_{ij} d\Omega, \]  
\[ B_{\alpha i} = \int_{\Omega} (\Phi_{\alpha i} \Phi_{\psi}) d\Omega, \]  
\[ A_{\alpha i \beta} = \int_{\Omega} (\Phi_{\alpha i} \Phi_{\beta}) d\Omega, \]  
\[ D_{\alpha i \beta} = \int_{\Omega} (\Phi_{\alpha i} \Phi_{\beta i}) d\Omega. \]

### 3.2 TEMPORAL DISCRETIZATION

The Crank-Nicolson method is applied to the temporal discretization. Central difference is used at time step \( t^{n+\frac{1}{2}} \):

\[ u_{i}^{n+\frac{1}{2}} = \frac{1}{2}(u_{i}^{n+1} + u_{i}^{n}) \]
The temporal discretization of the state equation is shown as follows.

\[
\begin{align*}
\rho M_{\alpha\beta} \frac{u_{\beta i}^{n+1} - u_{\beta i}^n}{\Delta t} + \rho K_{\alpha\beta\psi j} u_{\beta j}^{n+\frac{1}{2}} - cH_{\alpha\psi i} p_{\psi i}^{n+\frac{1}{2}} + S_{\alpha\beta\psi j} u_{\beta j}^{n+\frac{1}{2}} - k B_{\alpha\psi i} \phi_{\psi i}^{n+\frac{1}{2}} - \rho f_i &= 0, \\
M_{\alpha\beta} \frac{p_{\psi i}^{n+1} - p_{\psi i}^n}{\Delta t} + K_{\alpha\beta\psi j} u_{\beta j}^{n+\frac{1}{2}} + cH_{\alpha\psi i} u_{\psi i}^{n+\frac{1}{2}} &= 0, \\
M_{\alpha\beta} \frac{\phi_{\beta j}^{n+1} - \phi_{\beta j}^n}{\Delta t} - u_{\beta j}^{n+\frac{1}{2}} A_{\alpha\beta\psi j} \phi_{\psi j}^{n+\frac{1}{2}} + \gamma D_{\alpha\beta\psi j} \phi_{\psi j}^{n+\frac{1}{2}} &= 0,
\end{align*}
\]

where \( \Delta t \) is time increment.

4 NUMERICAL STUDIES

In numerical studies, cavity flow is analyzed for verification of the Navier-Stokes equation using acoustic velocity in case 1 and two phase fluids are analyzed in case 2. As numerical conditions, acoustic velocity \( c \), Reynolds number, diffusion coefficient \( \gamma \), concentration coefficient \( k \), and surface tension coefficient \( \sigma \) is 4.7, 1.0, 0.0001, 0.0001, and 0.00001 respectively. Time increment \( \Delta t \) is set to 0.01.

4.1 CA VITY FLOW

The computational domain and boundary conditions in case 1 are shown in Figure 4. In this area, three sides are fixed walls, and \( x \) direction velocity is given only in the upper side, and \( \rho \) is 1.0.

Finite element mesh is represented in Figure 5. The total number of nodes and elements are 2,601 and 5,000, respectively.
Figure 6-9 shows pressure and streamline in case 1. Pressure on the top-left corner is low and pressure on the top-right corner is high. The flow spirals at center of the area.
4.2 TWO PHASE FLUIDS

In case 2, two phase fluids are analyzed. The computational domain and initial conditions are shown in Figure 10.

In case 2, $\phi = 1.0$ in $\Omega_1$, $\phi = 0.0$ in $\Omega_2$ are taken as initial conditions. $\rho_{|\Omega_1}$, $\rho_{|\Omega_2}$ and $\phi$ are set to 1.20, 1.00 and 0.50.

Finite element mesh is represented in Figure 11. The total number of nodes and elements are 6,561 and 12,800, respectively.

Figures 11-14 show concentration in case 2. Figures 15-17 show velocity and density in case 2. Shape of fluid 1 approaches circle by the effect of the surface tension. Velocity works on the boundary $\Gamma$ for keeping shape of fluid 1. As time advances, velocity becomes small. Fluid 1 diffuses uniformly according as boundary $\Gamma$ changes.
5 CONCLUSION

In this study, the Navier-Stokes equation and advection-diffusion equation are used as the governing equations for moving boundary problem. The two equation system is alternately calculated to move values closer to true values. As a result, two phase fluids are able to be analyzed. Using acoustic velocity for the Navier-Stokes equation, analysis considering density change can be performed.

References