A Study on the Fluid-Structure Interaction using LSFEM

Sung-Ho Lee*, Sung-Kie Youn*, Jeoung-Heum Yeon* and Bo-Nan Jiang**

*Department of Mechanical Engineering, Korea Advenced Institute of Science and Technology, 373-1, Taejon, Korea

**Department of Mathematics and Statistics, Oakland University, Rochester, Michigan, USA

ABSTRACT

A study on the fluid-structure interaction using LSFEM is presented. To consider fluid-structure interaction, staggered scheme, which solves fluid and structure respectively by separate solvers in a predictor-corrector fashion, is used. To analyze fluid-structure interaction effectively, LSFEM is introduced in analyzing the fluid region. 2-D, incompressible and viscous fluid in the steady state is considered. The structure region is restricted in plane strain state including nonlinear behavior, and is analyzed by using commercial Galerkin FEM code ABAQUS. Also, a remeshing scheme for the fluid region using artificial spring elements is suggested.

This solution procedure is applied to the flow around a slender structure problem and microvalve problem. The results are verified by comparing with the analytical solutions.

1. INTRODUCTION

In the micro-mechanical system such as microvalve and micropump where the members are highly flexible, fluid motion affects deformation of the structure and also deformation of the structure affects fluid motion. In the case, fluid-structure interaction must be considered for the analysis of the performance of the device. In general, staggered scheme, which solves fluid and structure respectively by separate solvers in a predictor-corrector fashion, is used. However there are some difficulties in solving fluid region by traditional mixed Galerkin method. It often requires special treatments such as upwinding scheme or artificial damping, and also LBB condition between primal variables and dual variables must be satisfied. However, with least squares finite element method(LSFEM) used in solving fluid domain, special treatment will not be needed and there will be no LBB conditions to be satisfied. Also, discretized system matrix will be always positive definite. Therefore treating fluid region with LSFEM has many benefits.

There are many attempts to solve the fluid-structure interaction problems. Nomura et al.[1] solved fluid with streamline upwind Petrov-Galerkin method and modeled structure with mass-spring elements. Dyka et al.[2] and Jeans et al.[3] solved fluid with boundary element method and structure with the Galerkin method. Ulrich et al.[4] solved fluid and structure with the commercial Galerkin FEM code FLOTRAN and ANSYS.

There have been numerous research works on LSFEM. Jiang[5, 6] solved many types of fluid problem and Maxwell equation with LSFEM. Bochve et al.[7] and Cai et al.[8] solved Stokes equation. Cai et al.[9] and Siu et al.[10] applied LSFEM to an elasticity problem.

LSFEM and structure with Galerkin FEM. This algorithm will be applied to the flow around a slender structure problem and microvalve problem.

2. LSFEM

For a given linear boundary value problem, governing equations can be converted to first order differential equations by introducing state variables. Boundary conditions can also be converted to the algebraic equations between state variables and original variables.

Now let us consider the linear boundary value problem.

$$Au = f \quad \text{in} \quad \Omega \qquad (1)$$

$$Bu = g \quad \text{on} \quad \Gamma \qquad (2)$$

A is the first order partial differential operator and B is a boundary algebraic operator. Suppose that $f \in L_2(\Omega)$. An appropriate subspace V of the Hilbert space $L_2(\Omega)$ can be chosen as

$$V = \{v \in L_2(\Omega) \mid Bu = g \quad \text{on} \ \Gamma\} \qquad (3)$$

For an arbitrary trial function, $v \in V$, residual function is defined like as

$$R = Av - f \qquad (4)$$

In LSFEM, a minimizer of the squared L_2 norm of the residual $\|R\|_0^2$ is considered as a solution.

$$\|R\|_0^2 = \int_\Omega (Av - f)^2 d\Omega \equiv I(v) \quad v \in V \quad (5)$$

The minimizer of $I(v)$ can be obtained as follows

$$\lim_{t \to 0} \frac{d}{dt} I(u + tv) = 2\int_\Omega (Au)^T (Au - f) d\Omega = 0 \quad v \in V \quad (6)$$

In summary, the problem can be stated as:

Find $u \in V$ such that

$$B(u,v) = F(v), \quad \forall v \in V \qquad (7)$$

where

$$B(u,v) \equiv (Au, Av) = \int_\Omega (Au)^T (Av) d\Omega \qquad (8)$$

$$F(v) \equiv (f, Av) = \int_\Omega f^T (Av) d\Omega \qquad (9)$$

The bilinear form (8) is symmetric. For a well-posed problem (1), the operator A is bounded below. As a consequence, when discretized, (8) always lead to a symmetric positive-definite matrix.

3. FLUID-STRUCTURE INTERACTION ALGORITHM

In the fluid-structure interaction problem, fluid affects structure and also structure affects fluid. To consider these interactions a staggered scheme, which solves fluid and structure respectively by separate solvers in a predictor-corrector fashion, is used.

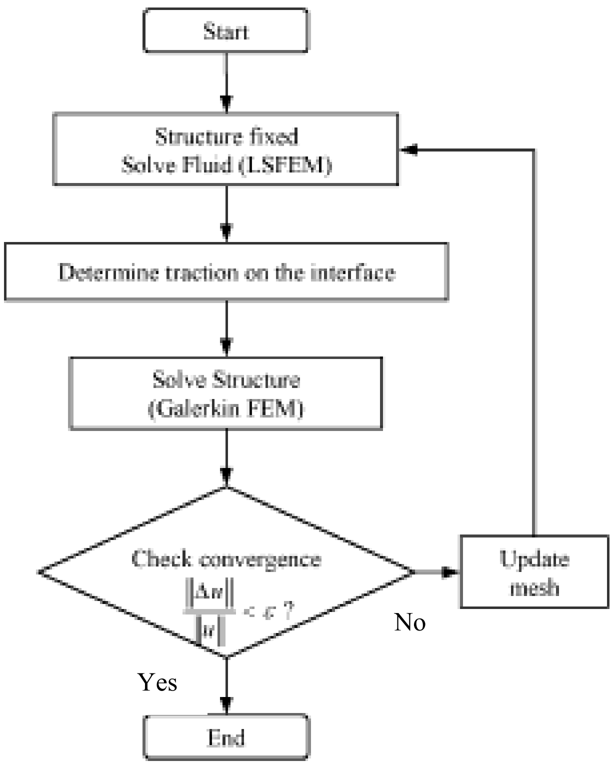


Fig.1 Flow chart of the solution procedure

as rigid. Thus, no-slip condition is applied at the fluid-structure interface and the traction can be calculated from the solution of fluid region. In solving the structure region, the displacement of structure can be obtained. Deformed shape of the structure changes the fluid domain. This affects the fluid behavior. The fluid region is remeshed for the next iteration. Flow chart of the solution procedure is in Fig.1.

The detailed description of each step is described below.

3.1. Fluid region

In this research, incompressible and viscous fluid in steady state is considered. The non-dimensionalized form of the governing equations are below.

$$v ? \; v + \; p - \frac{1}{\mathrm{Re}} \; {}^2 v = f \qquad (10)$$

$$?v = 0 \qquad (11)$$

Re is the Reynolds number and f is the non-dimensionalized body force. v and p are non-dimensionalized velocity and pressure. Since Eq.(10) is nonlinear, linearization is needed. In the successive substitution method, (10) is linearized as

$$v_0 ? \; v + \; p - \frac{1}{\mathrm{Re}} \; {}^2 v = f \qquad (12)$$

The subscript '0' in the above equation indicates that the value of the corresponding variable is known from the previous calculation step. This method has slow convergence but a large radius of convegence.

To make (11) and (12) in the first order form like (1), vorticity $\omega = \; \leftrightarrow v$ is introduced as a third variable.

Therefore, first order form of the governing equations are given as:

$$v_0 ? \; v + \; p + \frac{1}{\mathrm{Re}} \; \leftrightarrow \omega = f \qquad (13)$$

$$\omega - \; \leftrightarrow v = 0 \qquad (14)$$

$$?v = 0 \qquad (15)$$

2-D form of (13)-(15) in the Cartesian coordinate can be written as

$$\frac{f v_1}{f x} + \frac{f v_2}{f y} = 0 \qquad (16)$$

$$v_{10}\frac{f v_1}{f x} + v_{20}\frac{f v_1}{f y} + \frac{f p}{f x} + \frac{1}{\mathrm{Re}}\frac{f \omega}{f y} = f_x \qquad (17)$$

$$v_{10}\frac{f v_2}{f x} + v_{20}\frac{f v_2}{f y} + \frac{f p}{f y} - \frac{1}{\mathrm{Re}}\frac{f \omega}{f x} = f_y \qquad (18)$$

$$\omega - \frac{f v_1}{f y} - \frac{f v_2}{f x} = 0 \, . \qquad (19)$$

These equations can be written in simpler form:

$$A_1 \frac{f u}{f x} + A_2 \frac{f u}{f y} + A_o u = F \qquad (20)$$

where

$$u^T = \left[v_1 \quad v_2 \quad p \quad \omega \right]^T \qquad (21)$$

$$A_1 = \begin{bmatrix} 1 & 0 & 0 & 0 \\ v_{10} & 0 & 1 & 0 \\ 0 & v_{10} & 0 & -\frac{1}{\mathrm{Re}} \\ 0 & -1 & 0 & 0 \end{bmatrix} \qquad (22)$$

$$A_2 = \begin{bmatrix} 0 & 1 & 0 & 0 \\ v_{20} & 0 & 0 & \frac{1}{\mathrm{Re}} \\ 0 & v_{20} & 1 & 0 \\ 1 & 0 & 0 & 0 \end{bmatrix} \qquad (23)$$

$$A_0 = \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 \end{bmatrix} \qquad (24)$$

$$F^T = \left[0 \quad f_x \quad f_y \quad 0 \right]^T \qquad (25)$$

This first order type equation can be formulated with least squares method as explained in chapter 2.

3.2 Structure region

In this research, deformation of the structure is considered as a 2-D plane strain one with geometric nonlinearity due to the large deformation. It is realized using commercial Galerkin FEM code ABAQUS.

Traction boundary condition is applied on the fluid-structure interface. Displacement of the structure can be calculated. To check the convergence, an appropriate norm and convergence criterion are chosen as

$$\|u\|_C = \sqrt{\sum_{i=1}^{nnode} |u_i|^2} \qquad (26)$$

$$|u_i| = \sqrt{(u_i)_x^{\,2} + (u_i)_y^{\,2}} \qquad (27)$$

$$\frac{\|\Delta u\|_C}{\|u\|_C} < \varepsilon \qquad (28)$$

u is the total displacement, Δu incremental displacement current step, remeshing of the fluid region is needed.

3.3 Remeshing of the fluid region

Performing remeshing of the fluid region, distorted mesh leads to a bad numerical solution. Therefore an appropriate remeshing algorithm, which avoids distorted mesh, is needed.

This is realized using spring element shown in Fig.2. Finite elements in the fluid region are replaced with the spring elements. From the displacement solution of the structure region, deform the fluid-structure interface. From the resulting deformation of the spring elements in the fluid region, new mesh with minimum distortion can be obtained. In Fig.2, k_1 is related to the length of the each side of mesh and k_2 is related to the diagonal length of the mesh. Therefore k_1 is related to the volumetric deformation of the mesh and k_2 is related to the deviatoric deformation. Distortions in the region can be controlled by using appropriate combinations of values of k_1 and k_2.

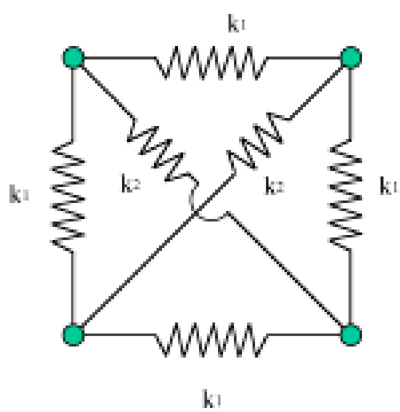


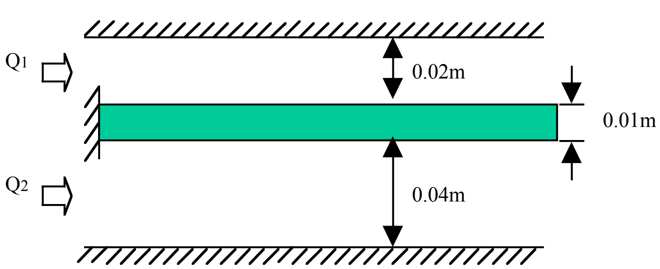
Fig.2 Artificial element for remeshing

4. EXAMPLES

4.1 Flow around a slender structure

Schematic diagram of the first example is in Fig.3. A channel is divided into two domains by clamped slender structure. The length of the structure is 0.75mm and thickness is 0.01mm. Fluid around the structure will cause pressure difference and it will deflect structure, and deflected structure will change the pressure difference. Therefore fluid-structure interaction occurs.

Flow rate of the upper and lower domain of the fluid respectively are $177\times10^{-6} m^2/s$ and $354\times10^{-6} m^2/s$. The flowing fluid is water with density $\rho = 1000 kg/m^3$, and viscosity $\mu = 0.001 kg/ms$ and it is modeled as incompressible viscous fluid. The structure is made of isotropic material with elastic modulus $E = 200GPa$, and poisson ratio $\nu = 0.3$.



When the structure is made of stiff material, deflection of the structure will be small. In this case we can obtain analytical solution for pressure difference between upper and lower surface of the structure and deflection of the structure. It is proposed by Wang[11].

Pressure difference between the upper and lower surface of the structure is presented in Fig.4. There are no visible difference between the analytical method and the proposed method which uses LSFEM. Deflection of the structure is presented in Fig.5. There are also no visible difference between the analytical method and the proposed method. These results show that the proposed method works well.

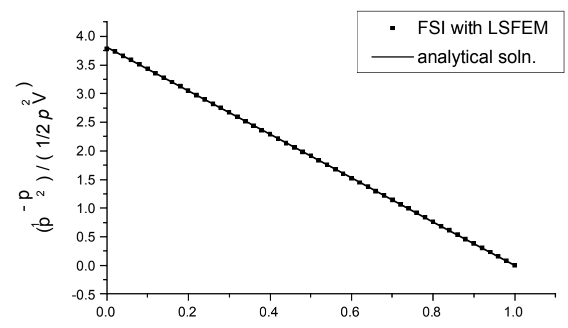


Fig. 4 Pressure difference between the upper and lower surface of the stucture

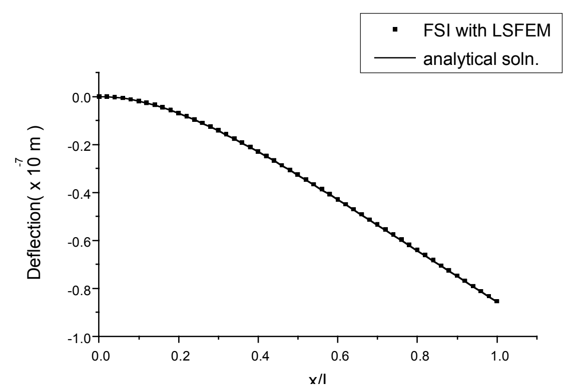


Fig. 5 Deflection of the slender structure

4.2 Flow simulation of a microvalve

4.2.1 Modeling

Schematic diagram of a microvalve is in Fig.6. The flap and valve seat in the microvalve has an important role. The flap consists of a thin plate with a length of 1700μm, a width of 1000μm and a thickness of 15μm. The valve seat has a squared form with a length of 400μm.

Top view of the microvalve is seen in Fig.7. Flow occurs through each side of valve seat. To reduce this 3-D problem to a 2-D problem, consider section AA' of Fig.7. AA' section of dashed box in Fig.6 is seen in Fig.8. The flow rate of this 2-D problem multiplied by total valve seat length 1600μm will be the flow rate of 3-D case.

Since the width and thickness ratio of the flap is very large, deformation of the flap can be modeled as plane

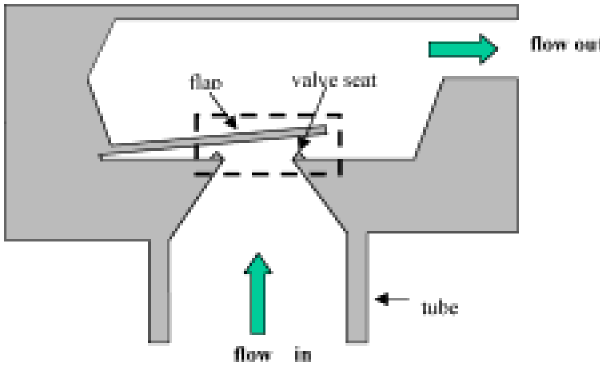


Fig. 6 Schematic diagram of the microvalve

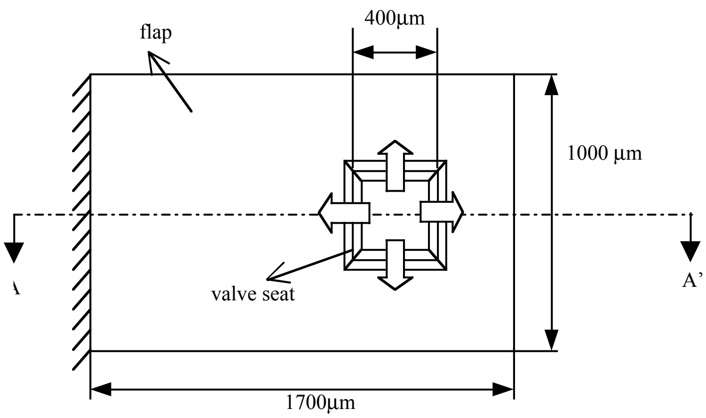


Fig. 7 Top view of the micro valve

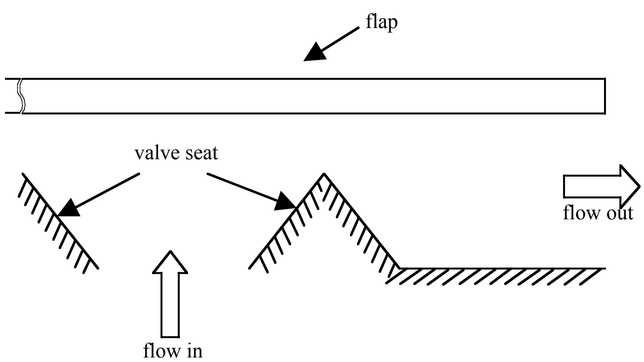


Fig. 8 Section AA' of Fig. 7

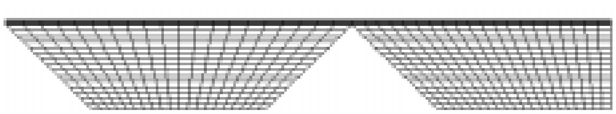


Fig. 9 Finite element grid of the fluid region in the microvalve

isotropic material with elastic modulus $E = 190GPa$, and poisson ratio $\nu = 0.25$. The working fluid is water with density $\rho = 1000kg/m^3$, and viscosity $\mu = 0.001kg/ms$ and it is modeled as an incompressible viscous fluid

4.2.2 Results

Fluid region is modeled using bilinear elements. It is seen in Fig.9. The flap is modeled using 8 node quadratic

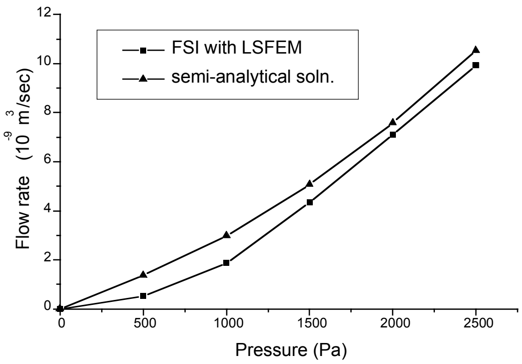


Fig. 10 Flow rate according to the working pressure

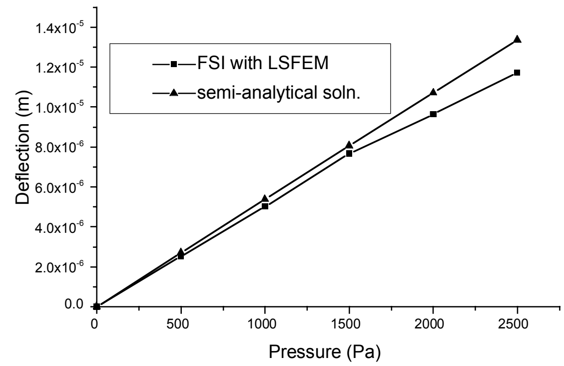


Fig. 11 Deflection of the flap according to the working pressure

Flow rate of the microvalve according to the valve seat pressure is calculated considering fluid-structure interaction. It is compared with the results of Ulrich et al.'s semi-analytical approach[4]. Ulrich et al. proposed a method to calculate the flow rate of microvalve. They solved fluid region with the extended Bernoulli equation which includes friction loss and solved flap motion using cantilever beam deflection equation. They solved iteratively each region and obtained converged solution.

Flow rate of the microvalve according to the valve seat pressure is presented in Fig.10. Flow rate increases as the valve seat pressure increases. Flow rate of the proposed method is lower than that of semi-analytical method. But, the tendency of the flow rate is similar in both cases. Deflection of the flap according to the valve seat pressure is presented in Fig.11. Deflection is nearly linear to the valve seat pressure in the semi-analytical method, but is not in the proposed method. Because nonlinear effect is considered in the proposed method but is not in the semi-analytical method. Deflection of the proposed method is lower than that of the semi-analytical method. It is compatible with the results of flow rates.

5. CONCLUSION

In this research, a robust method for the analysis of fluid-structure interaction in the micro-mechanical devices such as microvalve and micropump is presented. Making use of aforementioned advantage of LSFEM, many types of fluid problems can be analyzed without special treatment such as upwinding scheme or artificial damping. This method is verified by some examples incompressible and viscous flow is considered. Fluid region is solved using LSFEM. In the structure region, plane strain case is considered and solved using commercial Galerkin FEM code ABAQUS. Nonlinearities due to large deflection is considered.

As a first example, a flow around a slender structure is studied. The results from the present analysis were compared with analytical results. In the microvalve problem, flow rate according to the valve seat pressure is calculated. Reasonably accurate results are obtained from the proposed method.

As a future research, monolithic method for solving fluid-structure interaction problems is considered. By formulating structure problem also with LSFEM, a consistent monolithic method can be obtained.

REFERENCES

[1] T. Nomura and T. J. R. Hughes, "An arbitrary Lagrangian-Eulerian finite element method for interaction of fluid and a rigid body", Computer Methods in Applied Mechanics and Engineering, Vol. 95, pp. 115-138, 1992.

[2] C. T. Dyka, R. P. Ingel and G. C. Kirby, "Stabilizing the retarded potential method for transient fluid-structure interaction problem", International Journal for Numerical Methods in Engineering, Vol. 40, pp. 3767-3783, 1997.

[3] R. A. Jeans and I. C. Mathews, "Solution of fluid-structure interaction problems using a coupled finite element and variational boundary element technique", Journal of the Acoustical Society of America, Vol. 88, pp. 2459-2466, 1990.

[4] J. Ulrich and R. Zengerle, "Static and dynamic flow simulation of a KOH-etched microvalve using the finite-element method", Sensors and Acutators, Vol. 53, pp. 379-385, 1996.

[5] B.-N. Jiang, "The least-squares finite element method", Springer-Verlag, 1998.

[6] B.-N. Jiang, "On the least-squares method", Computer Methods in Applied Mechanics and Engineering, Vol. 152, pp. 239-257, 1998.

[7] P. B. Bochev and M. D. Gunzburger, "Least-squares methods for the velocity-pressure-stress formulation of the Stokes equations", Computer Methods in Applied Mechanics and Engineering, Vol. 126, pp. 267-287, 1995.

[8] Z. Cai, T. A. Manteuffel and S. F. McCormick, "First-order system least squares for the Stokes equations with application to linear elasticity", SIAM Journal of Numerical Analysis, Vol. 34, pp. 1727-1741, 1997.

[9] Z. Cai, T. A. Manteuffel, S. F. McCormick and S. V. Parter, "First-order system least squares for planar linear elasticity: pure traction problem", SIAM Journal of Numerical Analysis, Vol. 35, pp. 350-335, 1998.

[10] A. H. P. Siu and Y. K. Lee, "A three-dimensional least-squares finite element technuque for deformation analysis", International Journal for Numerical Method in Engineering, Vol. 40, pp. 4159-4182, 1997.

[11] X. Wang, "Analytical and computational approach for some fluid-structure interaction analysis", Computers and Structures, Vol. 72, pp. 423-433, 1999.