# A STABILIZED FINITE ELEMENT COMPUTATION OF FLOW AROUND OSCILLATING 2D BODIES

Hyung Taek Ahn[*1] and Raheel Rasool[2]

# 안정화된 유한요소법을 이용한 진동하는 2차원 물체 주위 유동해석

안 형 택,[*1] 라술 라힐[2]

*Numerical study of an oscillating body in incompressible fluid is performed. Stabilized finite element method comprising of Streamline-Upwind/Petrov-Galerkin (SUPG) and Pressure-Stabilizing/Petrov-Galerkin (PSPG) formulations for linear triangular elements was employed to solve the 2D incompressible Navier-Stokes equations whereas the motion of the body was considered by incorporating the arbitrary Langrangian-Eulerian(ALE) formulation. An algebraic moving mesh strategy is utilized for obtaining body conforming mesh deformation at each time step. Two tests cases, namely motion of a circular cylinder and of an airfoil in incompressible flow were analyzed. The model is first validated against the stationary cases and then the capability to handle moving boundaries is demonstrated.*

**Key Words :** Stabilized Finite Element, Arbitrary Langrangian-Eulerian(ALE), Vortex-Induced Vibration (VIV)

## 1. INTRODUCTION

Solution algorithm for incompressible Navier-Stokes equation has been evolving in several communities. Perhaps the first successful algorithm for the equation is the MAC algorithm of Los Alamos National Laboratory [1]. After the MAC algorithm main stream for solving the incompressible Navier-Stokes equation is based on velocity project and solution of Pressure Poisson equation, so-called segregated algorithm. On the other hand, advanced algorithm developed in the compressible flow community was also applied to the incompressible flow, such as the artificial compressibility method (ACM) [2-5]. In the Artificial compressibility method, momentum and continuity equations are fully coupled and fully implicit. In the ACM, The equation system is solved simultaneously to the next solution state, which makes the major difference from the segregated algorithms.

In this paper, we present the third category of the methods, namely stabilized Finite Element method [6-9]. Stabilized Finite Element method is also a method that couples momentum and continuity equations strongly, hence fully implicit formulation is possible. The method is a generalization of standard Galerkin Finite Element method that is popular in Solid mechanics. It includes addition stabilization terms that make test function space eventually different from the trial function space. Finite Element Methods (FEM) is also recognized as one of the methods whose dependency on the spatial mesh is found to be not very strong and its ability to handle meshes of high distortion is well known. Moreover, the effectiveness of FEM in solid mechanics is well established as a result of which it is proving to be a powerful tool in analyzing complex problems involving fluid-structure interaction (FSI).

FEM is a weighted residual based method, simplest of

1 정회원, 울산대학교 조선해양공학부
2 울산대학교 대학원 조선해양공학부
* TEL : 052) 259-2164
* Corresponding author, E-mail: htahn@ulsan.ac.kr

which is the Galerkin method which makes use of identical trial and weighting functions to solve the weak form of the governing equation. This method, when applied to solid mechanics problems, generates symmetric stiffness matrices which results in a stable numerical solution. However, when a similar formulation is applied to a fluid flow problem, the results are found to be characterized by spurious oscillations. This instability is mainly due to the non-symmetric nature of the convective term of the Navier-Stokes equation which results in spurious node-to-node oscillations primarily in the velocity field. This behaviors becomes more pronounced for convection-dominated problems, i.e. high Reynolds number flows.

Several stabilization techniques have been developed to stabilize the Galerkin formulation for problems involving convective transport. Of them, the Upwind based methods employ upwind differencing of the convective operator. Upwind differences though being inherently stable, however are only first order accurate. This methodology is analogous to the technique of adding artificial diffusion. The loss of accuracy from the upwind formulations is usually evident from the excessively dissipative nature of the solution. One efficient way to address this instability problem was proposed by Hughes and Brooks (Brooks 1982) in which they had proposed the addition of a term which is a function of the residual of the assumed solution over the governing differential equation. The method has found wide range application for the solution of Navier Stokes equation, both compressible and incompressible and its robustness for different flow-physics problems is well established. In this study we have employed this approach in the form of Streamline Upwind/Petrov-Galerkin (SUPG) (Brooks 1982) and the Pressure Stabilizing/Petrov-Galerkin (PSPG) (Tezduyaar 1990) formulation.

An attempt to extend the capabilities of the SUPG/PSPG formulation to problems involving moving boundaries is made in this study. In order to incorporate arbitrarily motion of the object, the stabilized Finite-Element formualtion is to be presented in the arbitrary Lagrangian-Eulerian (ALE) frame of reference. An efficient moving mesh strategy [10] is presented for retaining optimal mesh quality at every time moment, and their effectiveness is to be presented.

The outline of this paper is as follows: In the first section general problem statement is made and the equation governing the flow characteristics is defined. The SUPG/PSPG based finite element flow solver is described and the mesh moving ALE algorithm is also defined in this section. Numerical results for the test cases of exterior flow around a cylinder and an airfoil are presented in the next section. Results for the cases where the obstacles are stationary are presented first followed by the results for moving obstacles. This is followed by the conclusion derived from these results and the future implementation of this methodology.

## 2. STABILIZED FINITE ELEMENT FORMULATION

### 2.1 INCOMPRESSIBLE NAVIER-STOKES EQUATIONS

Solution of incompressible Navier-Stokes equation is desired. At an instant $t \in (0,T)$ we consider a bounded region $\Omega_t$ in $R^{n_d}$, with boundary $\Gamma_t$, where $n_d$ is the number of spatial dimensions. Then the governing Navier-Stokes equation in incompressible form for the evaluation of the flow variable velocity $\boldsymbol{u}(\boldsymbol{x},t)$ and pressure $p(\boldsymbol{x},t)$ is given as,

$$\rho\left(\frac{\partial \boldsymbol{u}}{\partial t}+\boldsymbol{u}\cdot\nabla\boldsymbol{u}\right)-\nabla\boldsymbol{\sigma}=0 \quad on \quad \Omega_t \ \forall t\in(0,T) \qquad (1)$$

$$\nabla\cdot\boldsymbol{u}=0 \quad on \quad \Omega_t \ \forall t\in(0,T) \qquad (2)$$

where $\rho$ is the fluid density and $\sigma$ is the stress tensor which is given as

$$\boldsymbol{\sigma}=-p\boldsymbol{I}+\boldsymbol{T} \qquad (3)$$

with

$$\boldsymbol{T}=2\mu\boldsymbol{\epsilon}=\mu\left(\nabla\boldsymbol{u}+\nabla\boldsymbol{u}^T\right) \qquad (4)$$

where $\mu$ is the fluid viscosity. The boundary condition associated with this problem is given as

$$\boldsymbol{u}=\boldsymbol{g} \quad on \quad (\Gamma_t)_g \qquad (5)$$

$$\boldsymbol{n}\cdot\boldsymbol{\sigma}=\boldsymbol{h} \quad on \quad (\Gamma_t)_h \qquad (6)$$

where $(\Gamma_t)_g$ is the part of boundary associated with Dirichlet type and $(\Gamma_t)_h$ is with Neumann type.

### 2.2 STABILIZED FINITE ELEMENT FORMULATION

Suppose we have some suitably defined

finite-dimensional trial and test function space for the velocity and pressure: $S_{\boldsymbol{u}}^{h}$, $V_{\boldsymbol{u}}^{h}$, $S_{p}^{h}$, and $V_{p}^{h}=S_{p}^{h}$. The stabilized Finite Element formulation of the incompressible Navier-Stokes equations is given as follows: find $\boldsymbol{u}^{h}\in S_{\boldsymbol{u}}^{h}$ and $p^{h}\in S_{p}^{h}$ such that $\forall$ $\boldsymbol{w}^{h}\in V_{\boldsymbol{u}}^{h}$ and $p^{h}\in S_{p}^{h}\, q^{h}\in V_{p}^{h}$

$$\begin{aligned}
&\int_{\Omega}\boldsymbol{w}^{h}\cdot\rho\left(\frac{\partial\boldsymbol{u}^{h}}{\partial t}+\boldsymbol{u}^{h}\cdot\nabla\boldsymbol{u}^{h}\right)d\Omega\\
&-\int_{\Omega}\boldsymbol{\epsilon}(\boldsymbol{w}^{h}):\boldsymbol{\sigma}(p^{h},\boldsymbol{u}^{h})d\Omega\\
&-\int_{\Gamma_{h}}\boldsymbol{w}^{h}\cdot\boldsymbol{h}^{h}d\Gamma\\
&+\int_{\Omega}q^{h}\nabla\cdot\boldsymbol{u}^{h}d\Omega\\
&+\sum_{e=1}^{n_{el}}\int_{\Omega^{e}}\left(\tau_{S}\boldsymbol{u}^{h}\cdot\nabla\boldsymbol{w}^{h}+\tau_{P}\frac{1}{\rho}\nabla q^{h}\right)\\
&\qquad\cdot\left[\rho\left(\frac{\partial\boldsymbol{u}^{h}}{\partial t}+\boldsymbol{u}^{h}\cdot\nabla\boldsymbol{u}^{h}\right)-\nabla\cdot\sigma\right]d\Omega
\end{aligned}\tag{7}$$

where $\tau_{S}$ and $\tau_{P}$ are the stabilization parameters for momentum and continuity equations.

The ALE formulation is conveniently incorporated into the stabilized finite element formulation by replacing the convective velocity which is the relative velocity between the material and the mesh. The formulations leads to a system of $(n_{d}+1)$ simultaneous equations corresponding to the three unknowns (u, v and p) for each nodes in two dimensional coordinate system. In case of triangular finite elements, as used in this study, 9x9 matrix system is obtained for each individual element. The integration over time was performed using the Crank-Nicolson scheme to move to the next time level.

## 3. FLOW AROUND STATIONARY OBSTACLES

### 3.1 FLOW PAST A FIXED CIRCULAR CYLINDER

External flow around a circular cylinder is a well studied problem and provides suitable validation parameters for Navier Stokes solvers. For low Reynolds number i.e. less than 40, the flow is found to be steady and symmetrical about the vortex centre-line. However, beyond this Reynolds number the flow is characterized by vortex shedding in the form of a laminar vortex street which gradually transitions to turbulent if the Reynolds number is increased. Numerical simulations for flow past a circular cylinder were carried out at Reynolds number 1000. The flow in this regime is characterized by the separation of laminar boundary layer on the cylinder body with a turbulent wake. Periodic Karman vortex shedding is typically expected in this regime. Fig. 1 depicts the problem geometry and the associated boundary conditions for this problem. The domain was meshed with P1/P1 triangular elements as shown in Fig. 2.

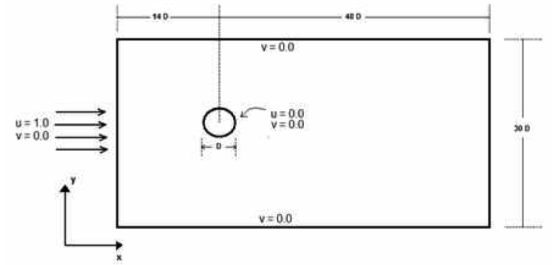


Fig. 1 Computational domain with associated boundary conditions

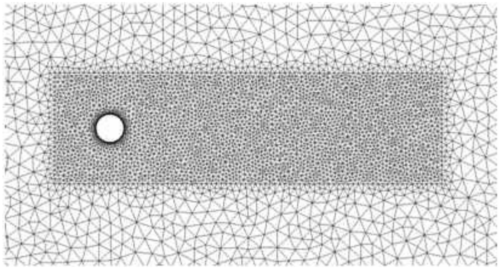

Fig. 2 Close-up view of the associated spatial mesh for the cylinder case

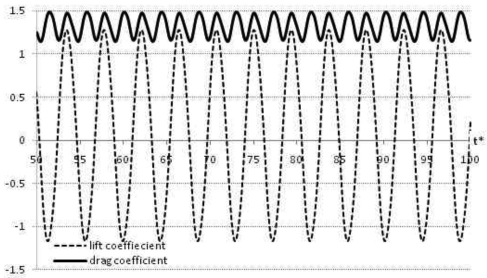


Fig. 3 Lift and drag coefficient for stationary cylinder at Re=1000

Fig. 3 represents the lift and drag force time histories for Reynolds number 1000 against the non-dimensional

Table 1 Drag coefficient comparison study for stationary cylinder [11-13]

| | $C_D$ |
|---|---|
| (Henderson 1995) | 1.51 |
| (Mittal 2008) | 1.45 |
| (Braza 1986) | 1.15 |
| Present | 1.314 |

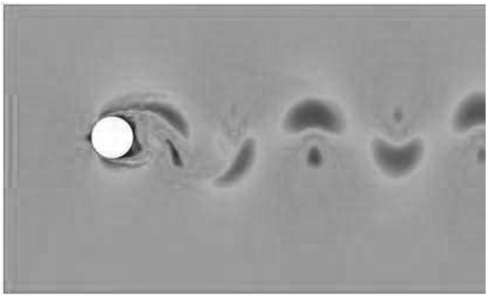

Fig. 4 Velocity contour plot for the stationary cylinder at Re=1000

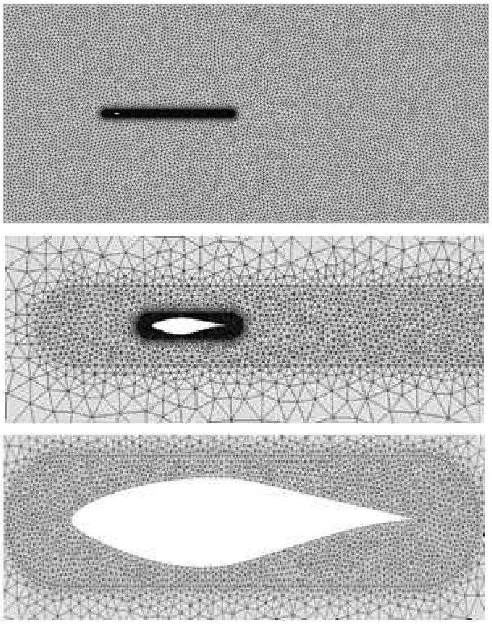

Fig. 5 Finite element mesh around S809 airfoil

time . As expected the nature of lift history is periodic about the centre line as a result f the periodic vortex shedding . Corresponding to this shedding frequency, the Strouhal number is calculated to be equal to 0.25. The mean drag coefficient was found to be 1.314. These results are found to be in suitable agreement with other established available data as suggested by Table 1.

Mesh and domain independency was also carried out by comparing the results of domains of different dimensions (16D, 24D and 30D in the vertical direction). The results of 30D domain are reported here. Fig. 4 provides a typical velocity contour plot once the vortex shedding has become periodic.

### 3.2 Flow past a Fixed Airfoil

Numerical simulations for flow past an airfoil was

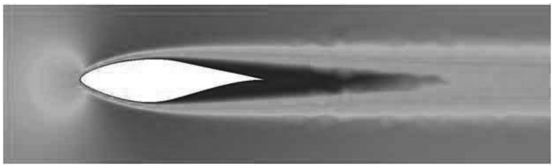

Fig. 6 Velocity contours for stationary airfoil at Re = 1000

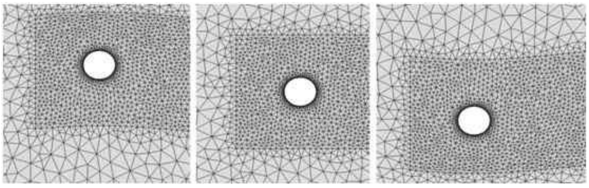

Fig. 7 Mesh deformation with translation of the circular cylinder along the cross-flow (vertical) direction

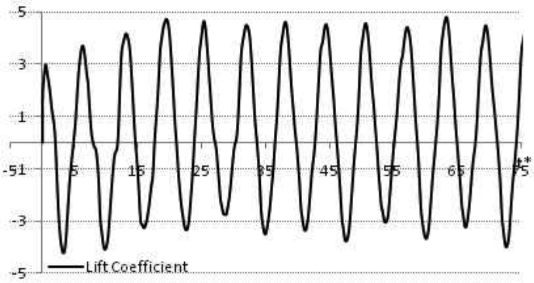


Fig. 8 Lift coefficient history for the oscillating cylinder at Re = 1000.

carried out. S809 airfoil was used for this purpose. The extent and the nature of the spatial domain were kept same as that of the previous case of the cylinder described in Fig. 1 Fig. 5 provides an illustration for the mesh used to analyze this problem. The mesh in the immediate vicinity of the airfoil was kept intensely dense in an effort to resolve the boundary layer. This can further be improved with the use of anisotropic triangular elements. The analysis was carried out at Reynolds number 1000 based on the chord length with the airfoil at zero angle of attack.

A time step of size 0.01 was used and results were considered once the flow had become uniform and the resulting forces had stabilized. The drag coefficient was found to be 0.134 while the lift coefficient was discovered to be 0.037. Velocity contours for this case are shown in Fig. 6.

## 4. Flow around Moving Obstacles

The modified SUPG/PSPG formulation with the incorporated ALE scheme was employed to the test cases

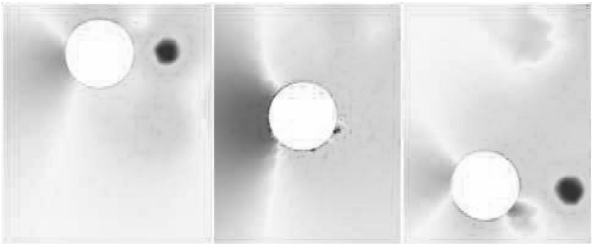

Fig. 9 Pressure contours plots during the oscillatory cycle of the cylinder.

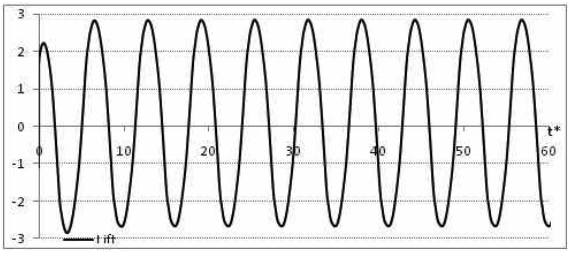


Fig. 10 Lift coefficient history for oscillating foil case at Re = 1000

discussed in the preceding section. Same mesh as that used in the pure Eulerian case was used to start the simulation. The mesh was then updated at each time-step to adjust to the motion of the obstacle. The mesh deformation was regulated by using an efficient algebraic scheme (Ahn 2009) [10]. In this scheme, the domain was divided into three sub-domains. The region in the immediate vicinity of the moving body moved rigidly with the body, while the region farthest away from the body remained fixed. The mesh motion is damped out in the middle, buffer region through a weighting parameter which is a function of the distance from the centre of the moving body to the mesh points under consideration. In this analysis, predefined rigid body motion in cross flow (y) direction was considered. For both the cases, the rigid-body motion was defined as a function oscillating in time along the vertical axis.

### 4.1 Flow past a Oscillating Circular Cylinder

The maximum amplitude for the translating of the body was 1 units away from the mean position. Mesh motion of oscillating cylinder is displayed in Fig. 7. The resulting time history of the lift force coefficient for the oscillating cylinder is shown in Fig. 8. The interaction of multiple frequencies is evident from this plot, one being contributed by the translation motion of the body while the high frequency component is due to the vortex shedding from the cylinder. Fig. 9 provides a screen shots of the

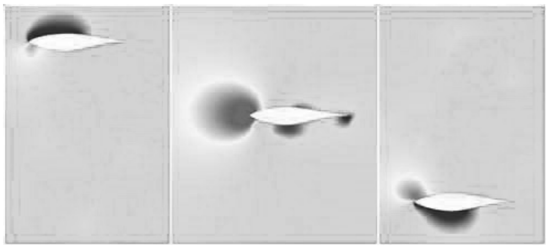

Fig. 11 Pressure contours plots during the oscillatory cycle of the S809 airfoil

pressure field around the cylinder during one of its oscillating period.

### 4.2 Flow past an Oscillating Airfoil

Same results for the oscillating S809 are given in Figure 10 and Figure 11. $(C_L)_{mean}$ for the airfoil case appears to be zero, this is consistent with the results obtained earlier for stationary cases where the lift coefficient for zero degree angle of attack was found to be very small. Hence in this case, the dominant contribution to the lift force is due to the oscillatory motion of the foil.

## 5. Conclusions

Stabilized finite element method based on SUPG/PSPG formulation was used in conjunction with arbitrary Langrangian Eulerian formulation to analyze exterior flow around moving rigid obstacles. The approach was applied to incompressible viscous flow problems using triangular finite elements with linear interpolation functions. The ALE scheme was conveniently adopted to the finite element formulation and resulted in effective representation of flow properties as governed by moving boundaries. The formulation was first validated against the established case of a stationary cylinder immersed in incompressible fluid. The capability to handle moving boundary was then demonstrated for the oscillating cylinder and oscillating airfoil. Future intent is to apply the stabilized finite element formulation with the ALE scheme to three dimensional rotating objects typically wind turbine blades.

## Acknowledgement

This work was supported by National Research Foundation of Korea Grant funded by the Korean Government (2009-0065381).

## REFERENCES

[1] 1965, Harlow, F. and Welch, J.E., "Numerical calculation of time-dependent viscous incompressible flow of fluid with a free surface." *Phys. Fluids*, Vol. 8, pp.2182-2189.

[2] 1967, Chorin, A.J., "A numerical method for solving incompressible viscous flow problems," *Journal of Computational Physics*, Vol.2, pp.12-26.

[3] 1990, Rogers, S.E. and Kwak, D., "Upwind differencing scheme for the time-accurate incompressible Navier–Stokes equations," *AIAA Journal*, Vol.28, pp.253-262.

[4] 2006, Ahn, H.T. and Kallinderis, Y., "Strongly coupled flow/structure interactions with a geometrically conservative ALE scheme on general hybrid meshes," J*ournal of Computational Physics*, Vol.219, pp. 671-696

[5] 2005, Kallinderis, Y. and Ahn, H.T., "Incompressible Navier–Stokes method with general hybrid meshes," *Journal of Computational Physics*, Vol.210, pp.75-108

[6] 1982, Brooks, A.N., Hughes, T.J.R., "Streamline Upwind/Petrov-Galerkin formulation for convection dominated flows with particular emphasis on the incompressible Navier Stokes equations," *Computer Methods in App. Mechs. and Engg.*, Vol.32, pp. 199-259.

[7] 1992, Tezduyar, T.E., "Stabilized finite element formulations for incompressible flow computations," *Advances in App. Mechs*, Vol.28, pp.1-44

[8] 1992, Tezduyar, T.E., Behr, M., "A new strategy for finite element computations involving moving oundaries and interfaces the Deforming Spatial Domain/Space-Time Procedure: I. The concept and the preliminary numerical tests," C*omputer Methods in App. Mech. and Eng*., Vol.94, pp.339-351.

[9] 2003, Tezduyar, T.E. and Sathe, S., "Stabilization parameters in SUPG and PSPG formulations," *J. Comp. and App. Mech*, Vol.4, pp 71-88.

[10] 2009, Ahn, H.T., Branets, L. and Carey, G.F., "Moving boundary simulations with dynamic mesh smoothing," *J. Numerical Methods in Fluids*.

[11] 1995, Henderson, R.D., "Details of the drag curve near the onset of vortex shedding," *Phys. of Fluids* Vol.7, pp.2102-2104.

[12] 2008, Mittal, R., Dong, H., Bozkurttas, M., Najjar, F.M., Vargas, A., and von Loebbecke, A., "A versatile sharp interface immersed boundary method for incompressible flows with complex boundaries," *Journal of Comp. Phys* Vol.227, pp 4825-4852.

[13] 1986, Braza, M., Chassaing, P. and Minh, H.H., "The Numerical Study and Physical Analysis of the Pressure and Velocity Fields in the Near Wake of a Circular Cylinder," *J. Fluid Mech*, Vol.165, p79.