Proceedings of the International Association for Shell and Spatial Structures (IASS) Symposium 2009, Valencia
Evolution and Trends in Design, Analysis and Construction of Shell and Spatial Structures
28 September – 2 October 2009, Universidad Politecnica de Valencia, Spain
Alberto DOMINGO and Carlos LAZARO (eds.)

Large eddy simulation based on stabilized finite element method for the analysis of wind field around spatial structures

Dai ZHOU*, Cheng HUANG[a]

* Professor, School of Naval Architecture, Ocean and Civil engineering,
Shanghai Jiao Tong University
No. 800 Dongchuan Road, Shanghai 200240, People's Republic of China
Email: zhoudai@sjtu.edu.cn

[a] School of Naval Architecture, Ocean and Civil Engineering,
Shanghai Jiao Tong University

Abstract

This paper presents a stabilized finite element formulation for large eddy simulation to predict the turbulent flow with high Reynolds number, which can be applied for the simulation of wind field and wind pressure distribution around shell and spatial structures. The Smagorinsky sub-grid scale model is applied for the governing equations of incompressible viscous flow and the streamline upwind Petrov-Galerkin (SUPG) weak formulation is adopted for momentum equation. For the spatial discretization, the same order iso-parametric interpolation process for the flow's velocity and pressure is introduced, and for temporal discretization, the explicit three-step finite element method is applied. All of those are adopted to overcome the instabilities of the FEM in solving the turbulent flow with high Reynolds number. For the numerical examples, we apply a lid driven flow of $Re=10^6$ in a square cavity and a flow past square cylinder at Re=22 000 with a moderate number of elements. Numerical results show that the present approach(that is, the combination of FEM with SUPG and the explicit three-step finite element method) can effectively suppress the computational instabilities for flow velocities and pressure fields, and is an efficient and reliable numerical procedure for solving turbulent flows with high Reynolds number.

Keywords: large eddy simulation, finite element method, Reynolds number, streamline upwind Petrov-Galerkin

1. Introduction

The understanding for the turbulent flow field with high Reynolds number is a seriously challenge for structural wind engineering. Nowadays, the prediction of wind flow around shell and spatial structures using computational fluid dynamics (CFD) is viable due to its high computational precise in describing the fluctuating motion of turbulence.

Among existing numerical techniques for predicting turbulent flows in CFD, large-eddy simulation (LES) appears to be one of the most promising approaches. In LES, the fluctuating motion of turbulence larger than the grid size can be computed exactly. Early LES technique is based on the Smagorinsky model(SM) for the unresolved sub-grid scales (SGS) (Smagorinsky [7]). Potential of this scheme has been clearly demonstrated in a turbulent channel flow by Deardorff [2]. Despite its limitation, it remains the most popular model due to its simplicity.

For the advantages of dealing with complex geometry and boundary conditions, finite element method (FEM) has been widely used for the solution of various fluid dynamic problems. However, it is known that classic Galerkin FEM meets great problems when applied to solve flow field with high Reynolds number. In order to overcome this drawback, some stabilized finite element formulations have been developed by many researchers. Among them, Streamline upwind/ Petrov-Galerkin (SUPG) method is famous which was proposed by Brooks and Hughes [1] and further developed by Hughes and Tezduyar [4,8].

In this paper, unsteady incompressible viscous flow with high Reynolds number is investigated and the formulation of Smagorinsky SGS model with SUPG stabilized term is applied to the computation of flow field. The same order iso-parametric interpolation for the flow's velocity and pressure is employed for the spatial discretization, the three-step finite element method is applied for the temporal discretization of the momentum equation and the Poisson pressure equation is derived from the incompressible condition. The above procedure on computing turbulent flow is conviniently used to predict the wind field around the buildings. On the other hand, for the verification of the present method, a series of the numerical examples of lid driven flow in a square cavity at Re=10^6 and flow past a square cylinder at Re=22 000 are carried out and some conclusions are discussed.

2. Governing equations

2.1. LES equations

The governing equations of LES turbulence model for incompressible viscous flow with SGS model proposed by Smagorinsky are as follows [7]:

$$u_{i,i} = 0 \tag{1}$$

$$u_{i,t} + u_j u_{i,j} = -p_{,i} / \rho + [(\nu + \nu_t)(u_{i,j} + u_{j,i})]_{,j} \tag{2}$$

where $v_t=(C_S h)^2(S_{ij}^2/2)^{0.5}$, $S_{ij}=(u_{i,j}+u_{j,i})/2$, $h=S^{1/2}$ for 2D, S is the area of 2D element. u and p are the velocity and pressure respectively, v is $1/Re$ as kinematic viscosity, v_t is the turbulent eddy viscosity, ρ is the fluid density. Constants used here, C_S=0.15 for 2D is suggested by Murakami [6].

2.2. Stabilized finite element formulations

The FEM weak form of the momentum equation can be actualized via Eq.(2) multiplying the velocity test function and added with SUPG stabilized term as follow [1],

$$\int_{\Omega}[\delta u_i(\frac{\partial u_i}{\partial t}+u_j u_{i,j})+\sigma_{ij}\delta u_{i,j}]d\Omega-\int_{\Gamma^h}\sigma_{ij}n_j\delta u_i d\Gamma+$$

$$\sum_{e=1}^{n_{el}}\int_{\Omega^e}\tau_{\mathrm{SUPG}}u_j\delta u_{i,j}(\frac{\partial u_i}{\partial t}+u_j u_{i,j}-\sigma_{ij,j})d\Omega=0 \qquad (3)$$

where n_{el} is the number of elements, δu_i is the velocity test function, n_j is the normal unit vector of the stress boundary Γ_h , σ_{ij} is the stress tensor given by

$$\sigma_{ij}=-p\delta_{ij}/\rho+(\nu+\nu_t)(u_{i,j}+u_{j,i}) \qquad (4)$$

and the stabilization parameter τ_{SUPG} is defined as follow(Dettmer and Peric′[3]),

$$\tau_{\mathrm{SUPG}}=\frac{\mathrm{h}^e}{2\|\mathrm{u}^e\|\rho}z \qquad z=\frac{\beta_1}{\sqrt{1+\left(\frac{\beta_1}{\beta_2\,\mathrm{Re}^e}\right)^2}} \qquad \mathrm{Re}^e=\frac{\|\mathrm{u}^e\|\mathrm{h}^e}{2(\nu+\nu_t)} \qquad (5)$$

where h^e, u^e and Re^e represent the characteristic element size, convective velocity and the Reynolds number of a element respectively. Here we use $(\nu+\nu_t)$ to describe the element viscosity. And β_1 define the limits of z as Re^e near to infinite and β_2 define the derivative dz/dRe^e at Re^e=0. In this work β_1=1 and β_2=1/3 have been obtained, and the characteristic element size is defined as the diameter of a circle which area is equal to the element area.

3. Computation formulations

3.1. Spatial discretization

The spatial discretization of Eq.(3) is performed using the finite element method and the same order interpolation is adopted for both flow velocity and pressure. Thus the trial function and the test function for both flow velocity and pressure could be expressed by means of Φ_I, and the flow velocity and pressure in an element can be described as follow,

$$u_i=\Phi_I u_{iI} \qquad p=\Phi_I p_I \qquad (6)$$

where u_{iI} is velocity at node I in the i direction, p_I is the pressure at node I.

Substituting Eq(6) into Eq(3), the finite element formulation for momentum equation is expressed as follow,

$$M_{IJ}\frac{\partial u_{iJ}}{\partial t}+N_{IJ}u_{iJ}-G_{iIJ}p_J+H_{iI}+D_{IJ}u_{iJ}-B_{iI}=0 \qquad (7)$$

where

$$M_{IJ} = \int_{\Omega} \Phi_I \Phi_J d\Omega + \tau_{SUPG} \int_{\Omega} \Phi_{I,j} \Phi_J \Phi_K u_{jK} d\Omega \tag{8}$$

$$N_{IJ} = \int_{\Omega} \Phi_I u_{iK} \Phi_K \Phi_{J,i} d\Omega + \tau_{SUPG} \int_{\Omega} \Phi_{I,j} \Phi_L u_{jL} \Phi_K u_{jK} \Phi_{jJ} d\Omega \tag{9}$$

$$G_{iIJ} = \frac{1}{\rho} \int_{\Omega} \Phi_{I,i} \Phi_J d\Omega - \tau_{SUPG} \frac{1}{\rho} \int_{\Omega} \Phi_{I,j} \Phi_K u_{jK} \Phi_{J,i} d\Omega \tag{10}$$

$$H_{iI} = (v + v_t) \int_{\Omega} \Phi_{I,j} \Phi_{J,i} u_{jJ} d\Omega \tag{11}$$

$$D_{IJ} = (v + v_t) \int_{\Omega} \Phi_{I,j} \Phi_{J,j} d\Omega \tag{12}$$

$$B_{iI} = \int_{\Gamma} \Phi_I \sigma_{ij} n_j d\Gamma \tag{13}$$

3.2. Temporal discretization

Explicit three-step finite element method is applied here for the temporal discretization of the momentum equation and the formulation of the momentum equation at each step is expressed as(Jiang and Kawahara[5]),

$$M_{IJ}^{n} \frac{u_{iJ}^{n+\frac{1}{3}} - u_{iJ}^{n}}{\Delta t/3} = -N_{IJ}^{n} u_{iJ}^{n} + G_{iIJ}^{n} p_J^{n} - H_{iI}^{n} - D_{IJ}^{n} u_{iJ}^{n} + B_{iI}^{n} \tag{14}$$

$$M_{IJ}^{n+\frac{1}{3}} \frac{u_{iJ}^{n+\frac{2}{3}} - u_{iJ}^{n+\frac{1}{3}}}{\Delta t/3} = -N_{IJ}^{n+\frac{1}{3}} u_{iJ}^{n+\frac{1}{3}} + G_{iIJ}^{n+\frac{1}{3}} p_J^{n} - H_{iI}^{n+\frac{1}{3}} - D_{IJ}^{n+\frac{1}{3}} u_{iJ}^{n+\frac{1}{3}} + B_{iI}^{n+\frac{1}{3}} \tag{15}$$

$$M_{IJ}^{n+\frac{2}{3}} \frac{u_{iJ}^{n+1} - u_{iJ}^{n+\frac{2}{3}}}{\Delta t/3} = -N_{IJ}^{n+\frac{2}{3}} u_{iJ}^{n+\frac{2}{3}} + G_{iIJ}^{n+\frac{2}{3}} p_J^{n+1} - H_{iI}^{n+\frac{2}{3}} - D_{IJ}^{n+\frac{2}{3}} u_{iJ}^{n+\frac{2}{3}} + B_{iI}^{n+\frac{2}{3}} \tag{16}$$

where *Δt* represents the length of time increment, the superscripts of n , $n+\frac{1}{3}$ and $n+\frac{2}{3}$ denote the sub-steps of each time increment respectively.

3.3. Finite element formulation for the pressure equation

Before entering the next step u_i^{n+1} from Eq.(16), the pressure p^{n+1} has to be confirmed. By taking the divergence of both sides of Eq.(2) and considering the incompressible condition of $u_{i\,i}^{n+1}$=0, we can get Poisson pressure equation as follow,

$$\frac{p_{,ii}^{n+1}}{\rho} = \frac{u_{i,i}^{n}}{\Delta t} - (u_j^{n+\frac{1}{2}} u_{i,j}^{n+\frac{1}{2}})_{,i} + (\nu + \nu_t)(u_{i,j}^{n+\frac{1}{2}} + u_{i,j}^{n+\frac{1}{2}})_{,ij} \tag{17}$$

where $u^{n+\frac{1}{2}} = (u^{n+\frac{1}{3}} + u^{n+\frac{2}{3}})/2$.

By using the trial function and test function of Galerkin finite element in Eq.(6) and considering ν_t evaluated at the center of each element for each sub-step, the final finite element formulation for pressure of p^{n+1} is,

$$S_{IJ} p_J^{n+1} = -Q_{iIJ} u_{iJ}^{n} - R_{iIJ} u_{iJ}^{n+\frac{1}{2}} - T_I \tag{18}$$

where

$$S_{IJ} = \frac{1}{\rho} \int_{\Omega} \Phi_{I,i} \Phi_{J,i} d\Omega \tag{19}$$

$$Q_{iIJ} = \frac{1}{\Delta t} \int_{\Omega} \Phi_I \Phi_{J,i} d\Omega \tag{20}$$

$$R_{iIJ} = \int_{\Omega} \Phi_{I,j} \Phi_J \Phi_{K,i} u_{jK}^{n+\frac{1}{2}} d\Omega \tag{21}$$

$$T_I = \frac{1}{\Delta t} \int_{\Gamma} \Phi_I (u_i^{n+1} - u_i^{n}) n_i d\Gamma \tag{22}$$

4. Numerical examples

4.1. Lid driven flow in a square cavity at Re=10^6

In this section, a lid driven flow of Re=10^6 in a square cavity is considered to verify the present method. A unit horizontal velocity (*U*) is prescribed on the top side of unit length (*H*), while the no-slip boundary condition is imposed on all other sides. The computational mesh is showed in Figure 1. The total of 3200 unstructured elements with refined mesh at the corners is generated to discretize the square cavity and the time increment of 0.01s is applied for the computation.

Figure 2 shows the streamline of fluid flow at 100s, Figure 3(a) and Figure 3(b) show the velocity vector and the vorticity in the square cavity at 100s respectively. From them we can see that there is one main eddy in the middle of the cavity and a few weak and disordered vortexes with scale larger than grid scale near corners of square cavity.

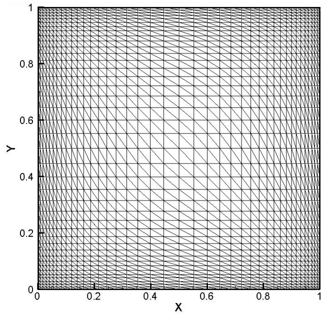


Figure 1: Finite element mesh

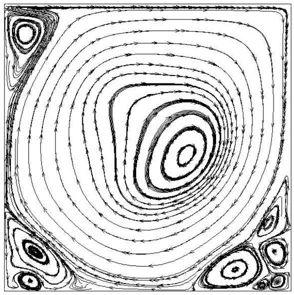
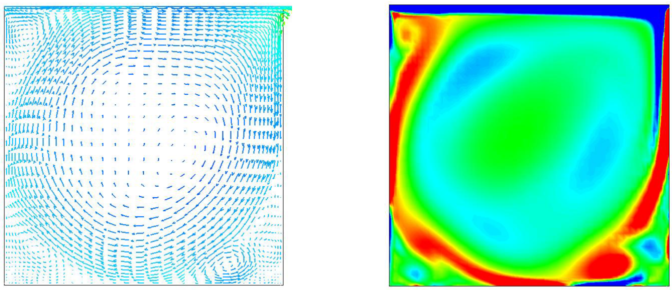


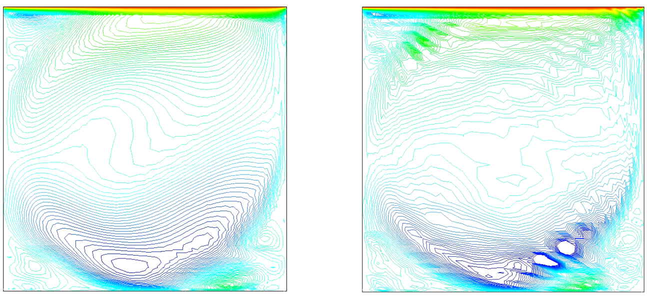
Figure 2: Streamline of fluid flow at 100s for Re=10^6



(a) velocity vector (b) vorticity

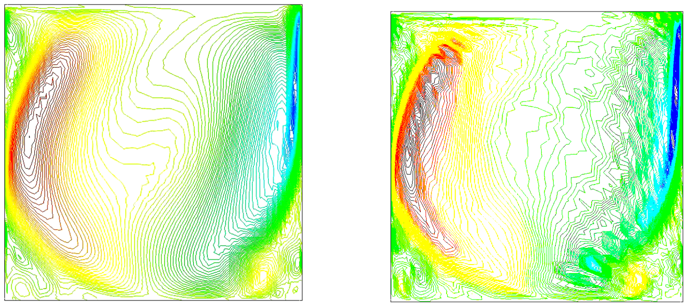
Figure 3: Velocity vector and vorticity of fluid flow at 100s for Re=10^6

Figure 4, Figure 5 and Figure 6 show the horizontal velocity (u) field, vertical velocity (v) field and pressure (p) field at 100s predicted by LES with and without SUPG stabilized term respectively. We can see that the present method has better stabilization for velocities and pressure fields than the normal LES technique. That is a strong demonstration of the stabilized effect of SUPG for finite element method.



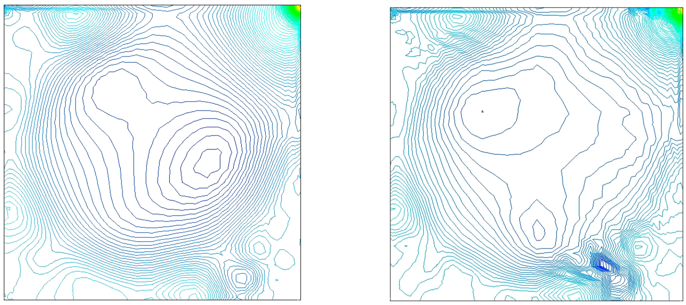
(a) LES with SUPG (b) LES without SUPG

Figure 4: Horizontal velocity (u) field at 100s for Re=10^6



(a) LES with SUPG (b) LES without SUPG

Figure 5: Vertical velocity (v) field at 100s for Re=10^6



(a) LES with SUPG (b) LES without SUPG

Figure 6: Pressure (p) field at 100s for Re=10^6

4.2. Turbulent flow past a square cylinder at Re=22 000

Two dimensional turbulent flow of Re=22 000 past a square cylinder is simulated. The computational region and element mesh are showed in Figure 7. The characteristic velocity (*U*) in streamwise direction from left inlet and the characteristic length (*H*) of the square cylinder are both unit value. The distances upstream and downstream in the square cylinder are 5*H* and 10*H* respectively. And the width of the region is 7*H*. Uniform mesh with 5027 rectangle elements is used to discretize the flow domain. The cylinder surface is no-slip boundary condition and the time increment is 0.01s.

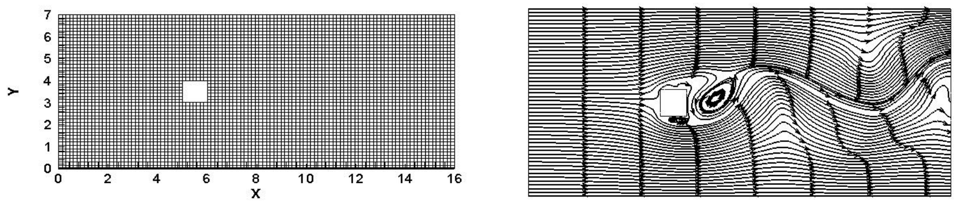
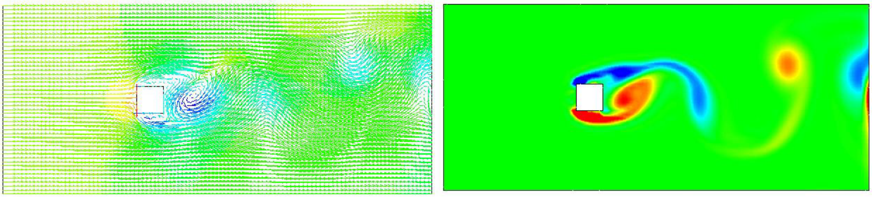


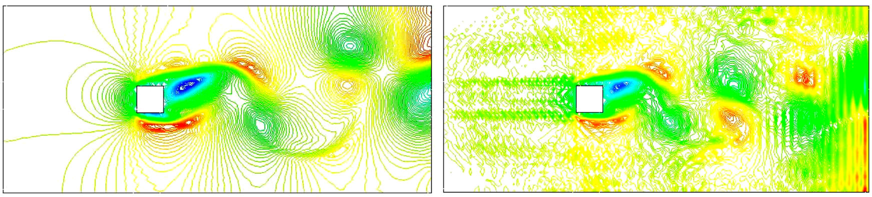
Figure 7: Geometry and mesh   Figure 8: Streamline of fluid flow at 100s for Re=22 000



(a) velocity vector   (b) vorticity

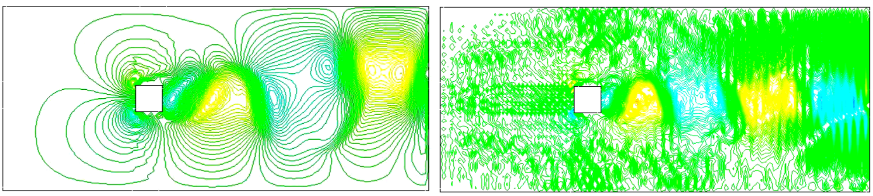
Figure 9: Velocity vector and vorticity of fluid flow at 100s for Re=22 000

Figure 8 shows the streamline of fluid flow at 100s, Figure 9(a) and Figure 9(b) show the velocity vector and the vorticity of the fluid flow at 100s respectively. We can see that there are obvious separation from the front corners and vortex street backwards from the square cylinder. Figure 10(a), Figure 11(a) and Figure 12(a) show the streamwise velocity (u) field, lateral velocity (v) field and pressure (p) field at 100s respectively which were computed by present method, while Figure 10(b), Figure 11(b) and Figure 12(b) show the streamwise velocity (u) field, lateral velocity (v) field and pressure (p) field at 100s respectively which were computed by normal LES finite element method without SUPG stabilized term. From them we can see that the present method has good stabilization and accuracy for both velocity and pressure fields without numerical oscillation even with a moderate number of elements.



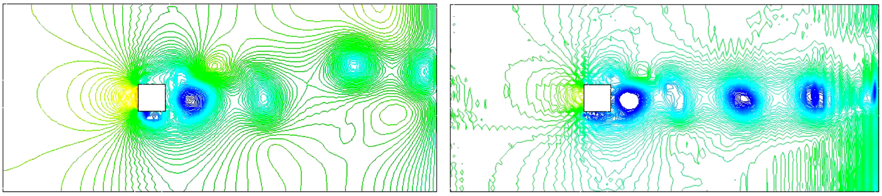
(a) LES with SUPG   (b) LES without SUPG

Figure 10: Streamwise velocity (u) field at 100s for Re=22 000



(a) LES with SUPG   (b) LES without SUPG

Figure 11: Lateral velocity (v) field at 100s for Re=22 000



(a) LES with SUPG (b) LES with SUPG

Figure 12: Pressure (p) field at 100s for Re=22 000

5. Conclusions

A stabilized finite element technique is developed to predict turbulent flow with high Reynolds number which can be applied for the simulation of wind field and wind pressure distribution around shell and spatial structures, where SUPG stabilized formulation is introduced. For the spatial discretization based on the finite element method, the same order iso-parametric interpolation process for the flow's velocity and pressure is adopted. And for the temporal discretization of the momentum equation, the explicit three-step finite element method is applied, which possesses second order temporal precise. Moreover, the flow pressure is investigated by the Poisson equation derived from the incompressible condition. On the other hand, the lid driven flow of $Re=10^6$ in a square cavity and turbulent flow past a square cylinder of Re=22000 are discussed respectively, as the numerical examples. Numerical results show that the present method can effectively suppress the computational instability in simulation of flow velocities and pressure field, and perfectly predict the turbulent flow with high Reynolds number.

Acknowledgement

Support from the Doctoral Disciplinary Special Research Project of Chinese Ministry of Education (No.200802480056) and the National Natural Science Foundation of China (Project No. 50778111) are acknowledged.

References

[1] Brooks A.N. and Hughes T.J.R., Streamline upwind/Petrov-Galerkin formulation for convection dominated flows with particular emphasis on the incompressible Navier-Stokes equations. *Journal of Computer Methods in Applied Mechanics and Engineering* , 1982; **32**; 199-259.

[2] J.W. Deardorff, A numerical study of three-dimensional turbulent channel flow at large Reynolds numbers. *Journal of Fluid mechanics,* 1970; **41**; 453-480.

[3] Dettmer W. and Peric′ D., A computational framework for fluid–structure interaction: Finite element formulation and applications. *Journal of Computer Methods in Applied Mechanics and Engineering* , 2006; **195**; 5754-5779.

[4] Hughes T.J.R., Franca L.P. and Balestra, M., A new finite element formulation for computational fluid dynamics: V. Circumventing the Babuška-Brezzi condition: A stable Petrov-Galerkin formulation of the Stokes problem accommodating equal-order interpolations. *Journal of Computer Methods in Applied Mechanics and Engineering* , 1986; **59**; 85-99.

[5] Jiang C. B. and Kawahara, M., A three step finite element method for unsteady incompressible flows. *Journal of Computational Mechanics*, 1993; **11**; 355-370.

[6] S.Murakami and A.Mochida, On turbulent vortex-shedding flow past 2D square cylinder predicted by CFD. *Journal of Wind Engineering and Industrial Aerodynamics* , 1995; **54/55**; 191-211.

[7] T.S. Smagorinsky, General circulation experiment with primitive equations: *Part I, Basic experiments, Monthly Weather Rev*, 1963; **91**; 99-164.

[8] Tezduyar T.E., Finite elements in fluids: Stabilized formulations and moving boundaries and interfaces. *Journal of Computers & Fluids* , 2007; **36**; 191-206.