Proceedings of the 10th Brazilian Congress of Thermal Sciences and Engineering -- ENCIT 2004
Braz. Soc. of Mechanical Sciences and Engineering -- ABCM, Rio de Janeiro, Brazil, Nov. 29 -- Dec. 03, 2004

Paper CIT04-0636

Large Eddy Simulation of Turbulent Incompressible Fluid Flows by a Nine-Nodes Control Volume-Finite Element Method

Rosiane Cristina de Lima (*)
(*) UNESP-Ilha Solteira, Av. Brasil, 56 – Ilha Solteira -SP
rosiane@aluno.feis.unesp.br

Sergio Said Mansur (*)
mansur@dem.feis.unesp.br

João Batista Campos Silva (*)
jbcampos@dem.feis.unesp.br

Abstract. The main purpose of this work is the numerical computation of incompressible fluid flows by a nine-node control volume finite element method (CVFEM) using the methodology of large-eddy simulation for turbulence modeling. The domain is discretized using nine-nodes finite elements and the equations are integrated into control volumes around the nodes of the finite elements. The Navier-Stokes equations are filtered for simulation of the large scale variables and the sub-grid scales stress appearing due to the filtering process are modeled through the eddy viscosity model of Smagorinsky. The two-dimensional benchmark problem of the lid-driven cavity flow is solved to validate the numerical code and the preliminary results are presented and compared with available results from the literature.

***Keywords**. Finite element, control volume, incompressible flow, large eddy simulation.*

1. Introduction

Flows of fluids are of interest in many engineering applications as well as in Nature. Most fluid flows are turbulent, occur in complex domains and are governed by a set of non-linear partial differential equations of convection-diffusive type. Therefore, practical solutions must be obtained through numerical techniques such as: The Finite Difference (FDM), the Finite-Volume (FVM) and/or the Finite Element Method (FEM). The finite volume method is the most popular method to calculate fluid flows and is characterized by terms in the equations with physical interpretation as fluxes, sources, etc., due to the conservative form of the equations. In the last decades, the finite element method, originally, developed for structural problems, has been improved for using in Computational Fluid Dynamics (CFD) constituting a powerful tool to simulate actual situations of fluid flows due to its versatility to discretizing complex geometry. Baliga & Patankar (1980) introduced a hybrid numerical method named control volume-finite element method. The control volume-finite element method (CVFEM) combines attractive characteristics from both the finite element and finite volume methods. In previous works, Campos_Silva & Moura (1997, 2001), Campos_Silva et al. (1998, 1999) and Campos_Silva (1998) presented a development of a control volume finite element method using a quadratic, quadrilateral nine-noded element to simulate unsteady, incompressible and viscous fluid flows. In those works no turbulence model was considered, so, the results were obtained for relatively low Reynolds numbers.

On the other hand, turbulent flows can be solved through direct numerical simulation (DNS) of the Navier-Stokes equations; either by using turbulence models (algebraic, two-equations models, second order models) with the Reynolds Average Navier-Stokes equations (RANS) or by simulating the large scales variables after filtering the Navier-Stokes equations and modeling the sub-grid-scales stresses in the called large-eddy simulation (LES) methodology. DNS is very expensive due the grid and time refinements required to capture all length and time scales of turbulence. According to Wilcox (1993) the number of grid points and time-steps for a DNS of a flow in a channel to reach a steady state is of order $6.7\mathrm{x}10^6$ points and 32,000 time-steps, for a Reynolds number of 12,300. Two-equation models are of more amenable costs and large eddy simulation occupies an intermediate position between DNS and two-equation models. In LES, for the same Reynolds of 12,300, it would be required about $6.1\mathrm{x}10^5$ grid points in a channel. With the increasing of the computational power, the use of LES to simulate fluid flows has gained many adepts.

In this work we present a large eddy simulation by using the CVFEM developed by Campos_Silva (1998). The benchmark problem of the lid-driven cavity flow is solved and the results are compared with results from the literature. After this introduction the article has been organized as follows: in section 2 we present the governing equations and some aspects of large-eddy simulation, in section 3 the main characteristics of the control volume-finite element method implemented are outlined, the results and some discussion are presented in section 4, in section 5 is presented conclusions, followed by acknowledgement in section 6 and references in section 7.

2. Analysis

In this section we present the mathematical model of the fluid flows and some aspects of the large-eddy simulation. Although the turbulence is a phenomenon of three-dimensional nature, in this step of the work we have considered only two-dimensional flows. Large-eddy simulations of three-dimensional flows by CVFEM are object of another study.

2.1. Governing equations

The unsteady, incompressible viscous Newtonian flows are governed by the momentum and the continuity equations (Navier-Stokes equations). In LES the equations are filtered for computation of the large-scale variables and the small scales computed are of size of the grid. There is a vast literature about turbulence and LES has been more and more used to simulate turbulent flows, mainly due to the increase of the power and speed of computers nowadays accessible for many researchers. It isn't our intent to do a full review of the literature on large eddy simulation in this work, the main goal here is the implementation of LES with a CVFEM of Campos_Silva (1998). Some excellent texts on LES were edited by Galperin & Orszag (1993). In Brasil, some groups at universities and research centers have published many works on the theme. In order to not to be unfair to none of them we preferred not try to list all papers from colleagues with much more experience in LES than us. A few works we have consulted were Matos, Pinho & Silveira_Neto (1999), Padilla & Silveira_Neto (2001), Almeida (2001) and Campregher (2002).

The filtered Navier-Stokes equations can be cast in the following general form for components of velocity u_i and pressure p:

$$\frac{\partial(\rho\bar{u}_i)}{\partial t^*} + \frac{\partial(\rho\bar{u}_j\bar{u}_i)}{\partial x_j} = -\frac{\partial\bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\mu\frac{\partial\bar{u}_i}{\partial x_j}\right) - \frac{\partial\tau_{ij}}{\partial x_j} + \bar{S}_i; \tag{1}$$

$$\frac{\partial\bar{u}_i}{\partial x_i} = 0\,. \tag{2}$$

where ρ and μ are the density and the dynamic viscosity respectively, S_i is a source term accounting for the other terms not appearing explicitly in Eq. (1).

In large-eddy simulation, the large-scale variables representing the velocity and pressure fields are defined by a filter function of the form:

$$\bar{f}(\vec{x}) = \int_D f(\vec{x}')G(\vec{x},\vec{x}',\Delta)d\vec{x}' \tag{3}$$

where an overbar denotes a filtered variable or large-scale variable on a domain *D*; *G* is a filter function, the most common are Gaussian, Top-hat or Sharp Fourier cut-off filters, Chidambaram (1998); and Δ is the filter width, generally, $\Delta = (\Delta_x\Delta_y\Delta_z)^{1/3}$ in 3D and $\Delta = (\Delta_x\Delta_y)^{1/2}$ in 2D with Δ_{x_i} being the grid size in the x_i-axis.

The sub-grid-scales (sgs) stress term τ_{ij} appearing in Eq (1) results from the filter process. This sub-grid-scales stress term τ_{ij} is defined as

$$\tau_{ij} = \rho\left(\overline{u_iu_j} - \bar{u}_i\bar{u}_j\right) \tag{4}$$

which must be modeled. In this work we have used the Smagorinsky model and the sgs stress term is modeled in the form

$$\tau_{ij} = \frac{2}{3}\rho k\delta_{ij} - 2\mu_t^*\bar{s}_{ij} \tag{5}$$

where the eddy viscosity, μ_t^*, the turbulent kinetic energy, k, and the deformation rate are defined by the following expressions, respectively

$$\mu_t^* = \rho (C_s \Delta)^2 (2 \bar{s}_{kl} \bar{s}_{kl})^{1/2}; \; k = \frac{\tau_{ii}}{2}; \; \bar{s}_{ij} = \frac{1}{2}\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right). \tag{6}$$

In the simulations of the present work the constant of Smagorinsky was set equal to $C_s^2 = 0{,}026$ and the filter width was chosen to be equal to the length of control volume face where the convective and diffusive fluxes are considered ($\Delta = \int_\xi \sqrt{(\partial x/\partial \xi)^2 + (\partial y/\partial \xi)^2}\, d\xi$ or $\Delta = \int_\eta \sqrt{(\partial x/\partial \eta)^2 + (\partial y/\partial \eta)^2}\, d\eta$). See Fig. 2 for reference.

After the substitution of Eqs. (5) and (6) into Eq. (1) one obtains the following equations, now, in dimensionless form:

$$\frac{\partial U_i}{\partial t} + \frac{\partial (U_j U_i)}{\partial X_j} = -\frac{\partial P_t}{\partial X_i} + \frac{\partial}{\partial X_j}\left(\left(\frac{1}{\mathrm{Re}} + \nu_t \right) \frac{\partial U_i}{\partial X_j} \right) + \frac{\partial}{\partial X_j}\left(\nu_t \frac{\partial U_j}{\partial X_i} \right) + F_i \tag{7}$$

$$\frac{\partial U_i}{\partial X_i} = 0 \tag{8}$$

where the term of kinetic energy was included in the pressure term resulting the turbulent pressure, $P_t = \bar{P} + \frac{2}{3}\frac{k}{u_0^2}$.

The dimensionless variables are defined as follows

$$X_i = \frac{x_i}{L}; \; U_i = \frac{\bar{u}_i}{u_0}; \; \bar{P} = \frac{\bar{p} - p_0}{\rho u_0^2}; \; t = \frac{t^*}{L/u_0}; \; \nu_t = \frac{\nu_t^*}{u_0 L} = \left(C_s \frac{\Delta}{L} \right)^2 (2 \bar{S}_{kl} \bar{S}_{kl})^{1/2}; \; \mathrm{Re} = \frac{\rho u_0 L}{\mu}. \tag{9}$$

3. Numerical Method

The CVFEM was firstly presented by Baliga and Patankar (1980, 1983) for triangular elements and later by Raw and Schneider (1986) for quadrilateral elements respectively. Several authors have enhanced the CVFEM since that time till nowadays. Raw et al. (1985) applied the nine-noded element to solve heat conduction problems. Banaszek (1989) did a comparison of the Galerkin and CVFEM methods in diffusion problems using six-noded and nine-noded elements. Campos_Silva (1998) developed a solver based on the nine-noded finite element and a control volume formulation to simulate 2D transient, incompressible, viscous fluid flows. During and after the development of the work of Campos_Silva (1998), some results were presented without considering the turbulence effect. Now, we have done simulations considering the model of Smagorinsky for the eddy viscosity. Campos_Silva (1998) and Campos_Silva & Moura (2001) presented the procedure of application of the CVFEM. The formulation of CVFEMs involves five basic steps, Saabas & Baliga (1994): (1) discretization of the domain of interest into elements; (2) further discretization of the domain into control volumes that surround the nodes in the finite element mesh, as shown in Figure 1; (3) definition of element-based interpolation functions for variables and physical properties of the fluid; (4) derivation of algebraic equations by using the sub-domain weighted residual method; and (5) assembling of the element equations forming the global matrix and choice of a procedure to solve the system of algebraic equations. Each node in the finite element mesh is inside a control volume. An element and its respective control volumes is showed in Figure 2, where are also showed the faces with convective and diffusive fluxes. These faces are identified as integration points.

The integration of the Eqs. (7) and (8) inside each of the nine control volumes in a finite element results

$$\int_V \frac{\partial U_i}{\partial t} dV + \oint_S \left(U_j U_i - \left(\frac{1}{\mathrm{Re}} + \nu_t \right) \frac{\partial U_i}{\partial X_j} \right) n_j dS + \int_V \frac{\partial P_t}{\partial X_i} dV = \oint_S \nu_t \frac{\partial U_j}{\partial X_i} n_j dS + \int_V F_i dV \tag{10}$$

$$\int_V \frac{\partial U_i}{\partial X_i} dV = 0 \tag{11}$$

with S and V denoting the surface area and the volume of a control volume around a node in the element, n_j is the outward normal vector to the area of a control volume where there are convective and diffusive fluxes, this vector normal has been defined as $\vec{n}dS = n_1 dS\vec{i} + n_2 dS\vec{j} = dy\vec{i} - dx\vec{j}$ for integration in the counterclockwise direction.

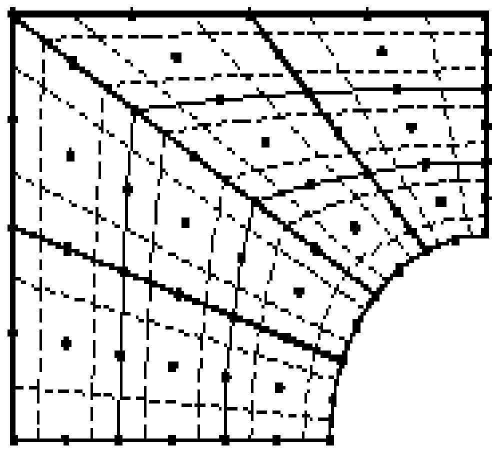

Figure 1. Meshes of finite elements and control volumes.

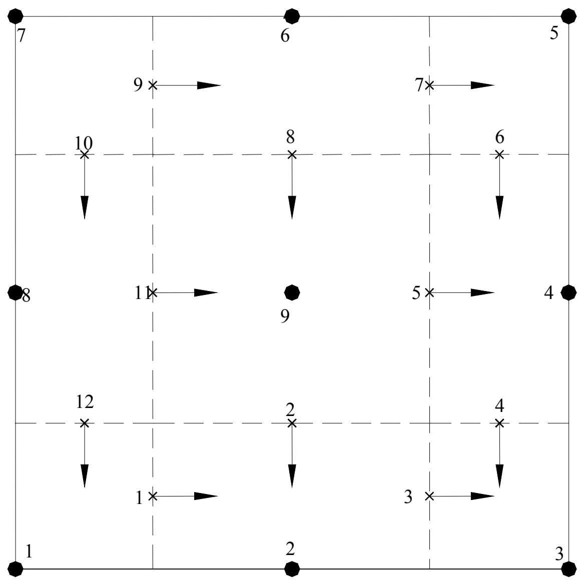

Figure 2. Element divided in control volumes and integration points

In order to transform the integrals of Eqs. (10) and (11) into algebraic equations, the variables in those integrals have to be interpolated by appropriated functions. We have interpolated the variables and coordinates by Lagrangian interpolating functions. The variables inside each element are then interpolated in the form

$$U^e(X(\xi,\eta),Y(\xi,\eta),t) = \sum_{\alpha=1}^{nnep} N_\alpha(\xi,\eta)U_\alpha(t) \tag{12}$$

$$V^e(X(\xi,\eta),Y(\xi,\eta),t) = \sum_{\alpha=1}^{nnep} N_\alpha(\xi,\eta)V_\alpha(t) \tag{13}$$

$$P^e(X(\xi,\eta),Y(\xi,\eta),t) = \sum_{\alpha=1}^{nnel} N_\alpha(\xi,\eta)P_\alpha(t) \tag{14}$$

$$X^e(\xi,\eta) = \sum_{\alpha=1}^{nnep} N_\alpha(\xi,\eta)X_\alpha \tag{15}$$

$$Y^e(\xi,\eta) = \sum_{\alpha=1}^{nnep} N_\alpha(\xi,\eta)Y_\alpha \tag{16}$$

where N_α are interpolating functions in master element, Figure 3b, (Dhatt & Touzot, 1984); U_α and V_α are the velocity components; P_α is the pressure, X_α and Y_α are the coordinates at nodes α of a element; nnep and nnel are numbers of nodes of quadratic (parabolic) and linear elements respectively. In order to satisfy the Ladyzhenskaya-Babuska-Brezzi (LBB) condition, the velocity field is interpolated by quadratic functions (nine-node elements) and the pressure is interpolated by linear functions (four-node elements) in a so called mixed formulation. After substitution of Eqs. (12)-(16) into Eqs. (10) and (11) results the algebraic equations

$$M_{\alpha\beta}\dot{U}_{i\beta} + C_{\alpha\beta}(U_{i\beta})U_{i\beta} - S_{i\alpha\beta}U_{i\beta} + H_{i\alpha\beta}P_\beta = F_{i\alpha} \tag{17}$$

$$D_{i\alpha\beta}U_{i\beta} = 0 \tag{18}$$

where $M_{\alpha\beta}$, $C_{\alpha\beta}$, $S_{i\alpha\beta}$, $H_{i\alpha\beta}$, $D_{i\alpha\beta}$ and $F_{i\beta}$ are coefficients of the mass, convection, diffusive, pressure term and continuity matrices and the source term vector, respectively, for each element. The definition for these matrices can be found in Campos_Silva (1998).

After the time discretization of Eqs. (17) and (18) one obtains the algebraic equations

$$\frac{M_{\alpha\beta}}{\Delta t}U_{i\beta}^{n+1} + \lambda(C_{\alpha\beta}^{n+1} - S_{i\alpha\beta})U_{i\beta}^{n+1} + \lambda H_{i\alpha\beta}P_\beta^{n+1} = F_{i\alpha}^n \tag{19}$$

$$D_{i\alpha\beta}U_{i\beta}^{n+1} = 0 \tag{20}$$

with the source term in Eq. (19) defined as

$$F_{i\alpha}^n = \frac{M_{\alpha\beta}}{\Delta t}U_{i\beta}^n - (1-\lambda)(C_{\alpha\beta}^n - S_{i\alpha\beta})U_{i\beta}^n - (1-\lambda)H_{i\alpha\beta}P_\beta^n \tag{21}$$

and the parameter of time discretization is in the range: $0 \le \lambda \le 1$. If fully implicit scheme is employed, $\lambda = 1$, and in this case there is no need of an initial condition for the pressure field.

In order to simplify the integration process each element in global coordinates, Figure 3a, has been mapped into a master element in local coordinates, Figure 3b. A further mapping must be done to each control volume area and contour inside elements by the using of the Gaussian quadrature rule of integration. Integrals in areas defined as $\int_{\xi_1}^{\varsigma_2}\int_{\eta_1}^{\eta_2} f(\xi,\eta)d\xi d\eta$ can be calculated as $\int_{-1}^{1}\int_{-1}^{1} g(r,s)drds$. Integrals in contours may be of the type $\int_{\xi_1}^{\varsigma_2} f(\xi)d\xi$ or $\int_{\eta_1}^{\eta_2} f(\eta)d\eta$ and can be calculated as $\int_{-1}^{1} g(s)ds$. In the range [-1,1] the Gaussian quadrature can be employed. In this way, the coefficient matrices are computed element by element in local coordinates shown in Figure 3b and a global system of equations is assembled like in the classical finite element method. We have used nine Gauss points in areas to exactly integrate the mass matrix and three Gauss points for integration along contours. After the assembly, the global system of algebraic equations in matrix form is as follow

$$K(U)U = F \tag{22}$$

in which can be seen that the global matrix K depends on the global vector of unknowns U due to the non-linearity of inertia terms. *S*uccessive substitution has been employed until the convergence was attained. Newton's method faster converging could have been chosen but it has a small radius of convergence than the successive approximation, Jiang (1998). The global system of equations has been solved by the frontal method, Taylor & Hughes (1981). In reality, the global system is never totally assembled in computer memory, which is a characteristic of the frontal method. So, personal computers may be used to solve the problems of interest. Because data have to be stored in hard disc, the solution of large problems by the frontal is time consuming.

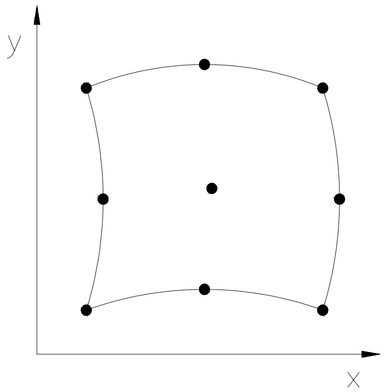

Figure 3.(a) - Element in global coordinates x-y

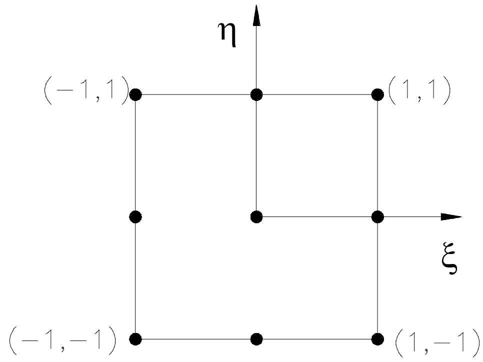

Figure 3.(b) - Element in local coordinates ξ-η

4. Results

Simulations have been done for the classical benchmark problem of the lid-driven cavity flow. However simple the geometry of the problem this flow present some interesting aspects like re-circulation zones that constitute a good challenge to numerical methods.

4.1. Lid-driven cavity

Flow in a square cavity induced by the upper plate motion has long been considered a benchmark problem, Figure 4. The velocities are set to zero at all walls, except the velocity along the x-axis for the upper plate. The pressure was set equal to zero at the mid bottom point. Figure 5 and 6 show the results for profiles of U(Y) velocity at X=0,5 (mid vertical line) and V(X) velocity at Y=0,5 (mid horizontal line) respectively for Reynolds numbers of 100, 400, 1,000 and 10,000 of the present work compared with results from Ghia, Ghia & Shin (1982), when the permanent regime is established. As can be seen on Figures 5 and 6 the agreement between our results and those from Ghia, Ghia & Shin(1982) is quite good, in spite of none upwind technique neither second order time integration have been used. The terms on the right hand side of Eq. (10) have been neglected in the simulations. The mesh for discretization of the domain was of 161 by 161 grid points with a total of 64964 variables. The mesh in this work was a few more refined than that of Ghia, Ghia & Shin, however, for low Reynolds numbers, even a coarse mesh has given results in good agreement with the ones from the literature. The stream functions are shown in Figures 7 to 9 for three Reynolds numbers of 400, 1,000 and 10,000 respectively. The integer numbers at right hand side of the cavity identify a stream function (SF) and the real numbers are the numerical values of the stream functions. The secondary vortices at corners of the cavity have been predicted satisfactorily. With the increase of the Reynolds number two secondary vortices are formed at right bottom corner and also one secondary vortex appears at left upper corner of the cavity. At left bottom corner the secondary vortex also increase in size with the increase of the Reynolds number.

5. Conclusions

In this work we have presented a large-eddy simulation by control volume–finite element method using nine-node finite elements with quadratic and linear interpolation functions for the velocity and pressure fields, respectively. The benchmark problem of the lid-driven square cavity has been solved in a Pentium 4, 2 GHz and 2 Gb of RAM personal computer. No upwind technique has been applied and a fully implicit method has been used for time discretization. The results for the cavity flow compare well with those ones from the literature. The solution of large problems is still slowly, but this can be improved with the use of more efficient techniques of solutions to linear systems. The natural sequence of this work is the three-dimensional simulations and the use of dynamic models to compute the constant of Smagorinsky. Some preliminary results in regions with inflow and outflow have been already obtained, encouraging the continuity of this work.

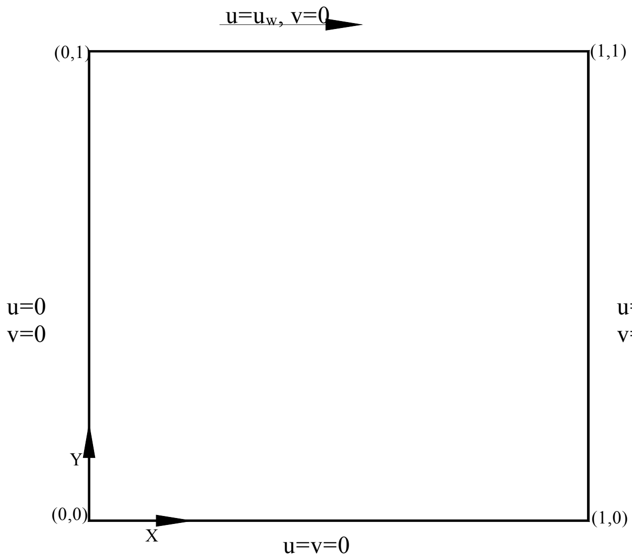

Figure 4. – Geometry and boundary conditions for the cavity flow

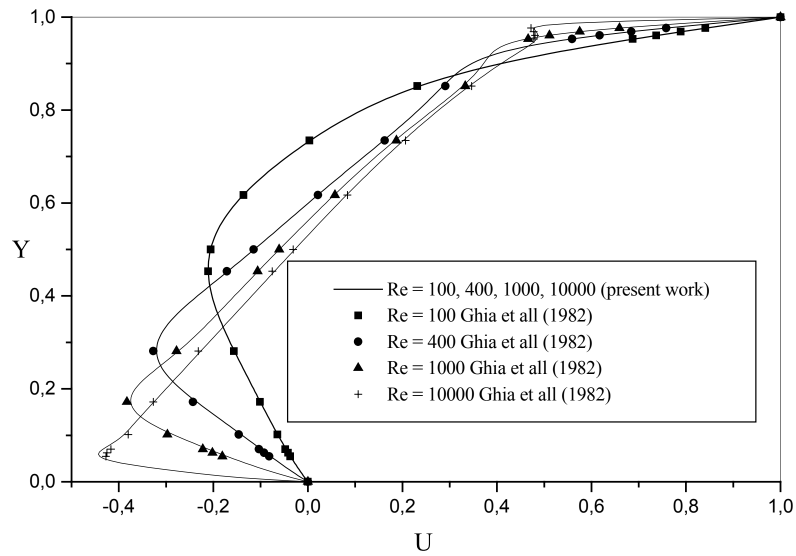

Figure 5. – U-velocity at X = 0,5

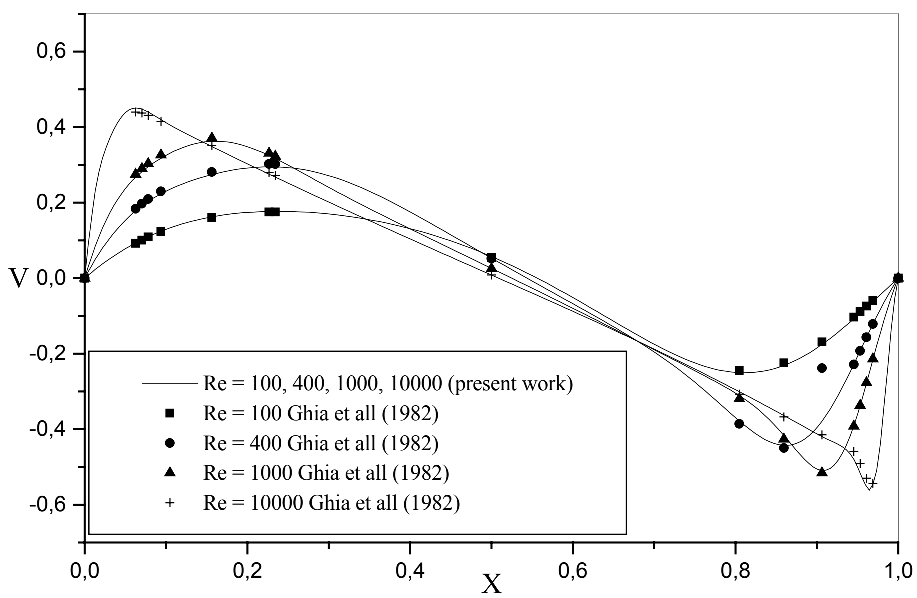

Figure 6. – V-velocity at Y = 0,5

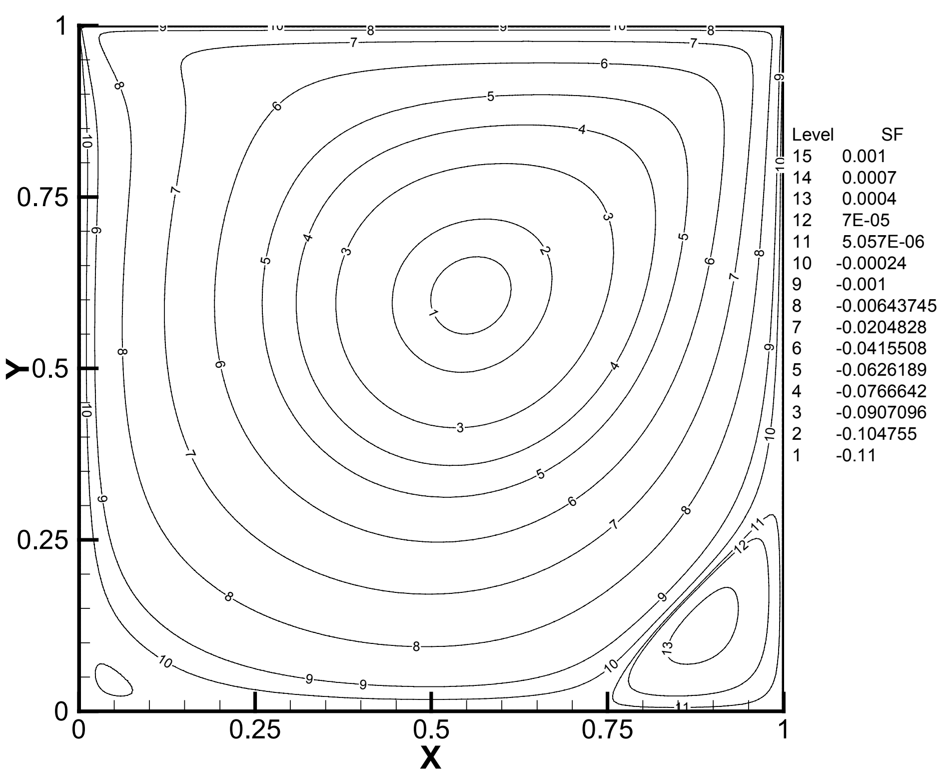

Figure 7. Stream function for Re = 400

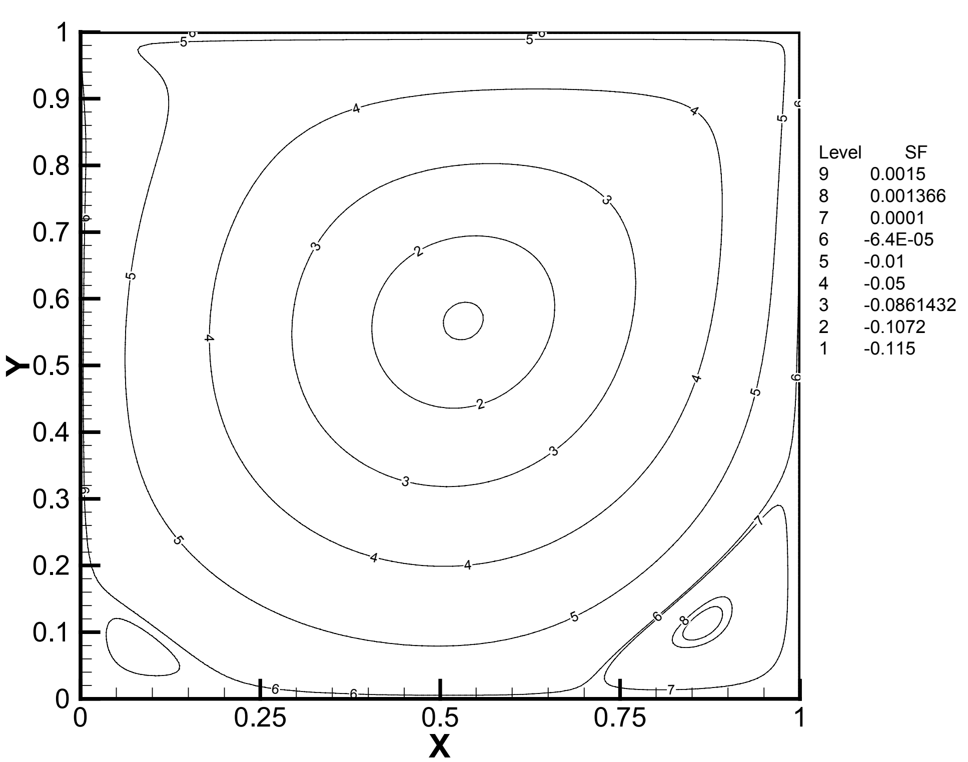

Figure 8. Stream function for Re = 1,000

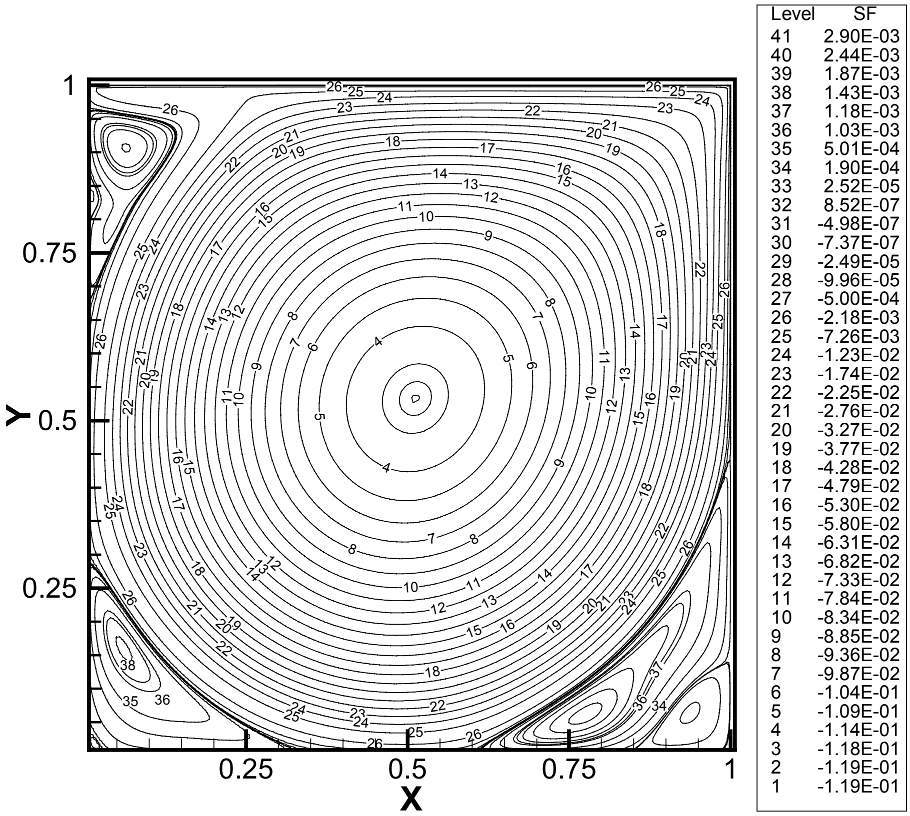

Figure 9. Stream function for Re = 10,000

6. Acknowledgement

The authors would like to acknowledge the FAPESP for the computational resources and the FUNDUNESP for the grants related to the development of this work.

7. References

Almeida., O., 2001, Numerical simulation of large scales of transitional flows around cylinders of square and rectangular base, Master Dissertation (in Portuguese), UNESP Ilha Solteira, Ilha Solteira, SP, Brazil.

Baliga, B.R. & Patankar, S.V., 1980, A new finite-element formulation for convection-diffusion problems, Numerical Heat Transfer, vol. 3., pp. 393-409.

Baliga, B.R. & Patankar, S.V., 1983, A control volume finite-element method for two-dimensional fluid flow and heat transfer, Numerical Heat Transfer, vol. 6, pp. 245-261.

Banaszek, J., 1989, Comparison of control volume and Galerkin finite element methods for diffusion-type problems, Numerical Heat Transfer, V. 16, p. 59-78.

Campos Silva, J.B. & Moura, L.F.M., 1997, Numerical simulation of fluid flow by the control volume-finite element method. Proceedings (in CD-ROM) of the XIV Brazilian Congress of Mechanical Engineering - COBEM97, December 8-12, Bauru-SP, Brazil, paper code 041, 8p.

Campos Silva, J.B., Vieira, E.D.R. & Moura, L.F.M., 1998, Control volume-finite element and flow visualization methods applied for unsteady viscous flow past a circular cylinder. Anais do V Congresso de Engenharia Mecânica do Norte-Nordeste, October 27-30, Fortaleza, CE, Brazil, Vol II, pp. 80-87.

Campos Silva, J.B., 1998, Numerical simulation of fluid flow by the finite element method based on control volumes, Doctorate Thesis (in Portuguese), State University of Campinas, Campinas, São Paulo, Brazil.

Campos Silva, J. B.; Aparecido, J. B. & Moura, L. F. M.. A Control Volume-Finite Element Method (CVFEM) for Unsteady Fluid Flows. Proceedings (in CD-ROM) of the XV Brazilian Congress of Mechanical Engineering - COBEM'99, Águas de Lindóia/SP, 22 a 26/11/99.

Campos Silva, J. B., & Moura, L. F. M. A Control-Volume-Finite-Element Method (CVFEM) for Unsteady, Incompressible, Viscous Fluid Flows. Numerical Heat Transfer Part B: Fundamentals Vol. 40 Nº 1, pp 61-82, 2001.

Campregher Junior, R. 2002, Numerical simulation of transitional and turbulent flows around Cartesian geometries, Master Dissertation (in Portuguese), UNESP Ilha Solteira, Ilha Solteira, SP, Brazil.

Chidambaram, N. (1998) Colocated-grid Finite Volume Formulation for the Large Eddy Simulation of Incompressible and Compressible Turbulent Flows. Graduate College, Department of Mechanical Engineering, Iowa State University, Ames, Iowa, USA. M.Sc. Thesis.

Dhatt, G. & Touzot, G., 1984, Une Presentation de la Méthode des Éléments Finis, Deuxieme Édition, Maloine S.A. Éditeur, Paris.

Guia, U., Guia, K.N. & Shin, C.T., 1982, High-Re Solutions for Incompressible Flow Using the Navier-Stokes Equations and a Multigrid Method, Journal of Computational Physics, 48, pp.387-411.

Jiang, B-N., 1998. The Least-Squares Finite Element Method: Theory and Applications in Computational Fluid Dynamics and Electromagnetics. Springer-Verlag.

Matos, A., Pinho, F.A.A. & Silveira_Neto, A., 1999, Large-eddy simulation of turbulent flow over a two-dimensional cavity with temperature fluctuations. International Journal of Heat and Mass Transfer, 42, pp. 49-59.

Padilla, E.L.M. & Silveira_Neto, A., 2001, Large-eddy simulation of transitional natural convection over a horizontal heated cylinder, using a dynamic sub-grid scale model. Proceedings of the 2nd International Conference on Computational Heat and Mass Transfer. Rio de Janeiro/SP, 22 a 26/10/2001.

Raw, M.J & Schneider, G.E., 1986, A skewed, positive influence coefficient up-winding procedure for control-volume-based finite-element convection-diffusion computation. Numerical Heat Transfer, vol. 9, pp. 1-26.

Raw, M.J., Schneider, G.E. & Hassani, V., 1985, A nine-noded quadratic control-volume-based finite element for heat conduction, J. Spacecraft, vol. 22, No. 5, pp.523-529.

Saabas, H.J. & Baliga, B.R., 1994, Co-located equal-order control-volume finite-element method for multidimensional, incompressible, fluid flow - Part I: Formulation, Numerical Heat Transfer, Part B, vol. 26, pp.381-407.

Saabas, H.J. & Baliga, B.R., 1994, Co-located equal-order control-volume finite-element method for multidimensional, incompressible, fluid flow - Part II: Verification, Numerical Heat Transfer, Part B, vol. 26, pp.409-424.

Taylor, C. & Hughes, T.G., 1981, Finite Element Programming of the Navier-Stokes Equations, Pineridge Press Ltd, Swansea, U.K.

8. Copyright Notice