APPLICATION OF LARGE EDDY SIMULATION TO SIMULATE TRANSITON AND TURBULENTE FLOWS IN LID-DRIVEN CAVITIES

Tadeu Tonheiro Rodrigues, tadeu.tonheiro@gmail.com
José Luiz Gasche, gasche@dem.feis.unesp.br
Departamento de Engenharia Mecânica, Faculdade de Engenharia de Ilha Solteira, Univerdade Estadual Paulista Júlio de Mesquita Filho, Av. Brasil, 56-Centro, 15385-000, Ilha Solteira, SP, Brasil

Abstract*. The numerical study of turbulent flows is one of the greatest current challenges on Computational Fluid Dynamics. In order to obtain the small turbulence scales appearing in those types of flow using Direct Numerical Simulation (DNS) for high Reynolds numbers, fine meshes must be used, turning DNS prohibitive in the current time. The Large Eddy Simulation is a promising methodology for the simulation of turbulent flows. It is based on splitting the turbulence scales through the application of a filter operator in the governing equations. After this operation, the greatest turbulence scales are directly simulated and the small turbulence scales are modeled using a sub-mesh model. The application of this methodology has generated good numerical results on the study of many fluid dynamics problems. Combining low computational cost and easy implementation it has become one of the most promising tools among the turbulence models. The objective of the present paper is to apply the Large Eddy Simulation methodology in a computational code based on the Finite Volume method to analyze the lid-driven cavity problem. The general features of the code are: applied to incompressible, two-dimensional and transient flows, the interpolation scheme for the convective-diffusive terms is the Power-law, the pressure-velocity coupling algorithm used is the SIMPLEC – SIMPLE Consistent – the transient term is discretised with first order Euler scheme, a staggered mesh for the velocity with respect to pressure is employed and the resulting equations systems are solved with the TDMA algorithm. The results were obtained for transitional and turbulent flows in a square two-dimensional cavity, with Reynolds numbers varying from 3200 to 10000, and were confronted with literature data showing good agreement.*

Keywords*: Large Eddy Simulation, Finite Volume Method, Lid-driven cavity.*

1. INTRODUCTION

The use of computational codes for simulation and studying of fluid mechanics and heat transfer problems is frequent on academic area and has gradually gained space on R&D area in industry. There are many commercial codes available in the market, like CFX and Fluent (ANSYS). As closed codes, sometimes is difficult to implement other mathematical physical modeling. Therefore, it is important to develop open codes in order to allow the implementation of new techniques and methodologies of simulation.

In this context, the study of transitional and turbulent flows is accompanied with a series of difficulties of computational order, due to the fact that the dimensions of the smallest turbulent eddies decreases with increasing Reynolds number, being necessary a computational grid quite refined to capture the smallest eddies, which increases the computational demand and conflicting with the current computational resources. Consequently, the development and implementation of turbulence models in open codes to simulate turbulent flows with the current computational resources is of fundamental importance. One of the most promising methodology is the Large Eddy Simulation (LES), whose elementary principle is the separation of the turbulence eddies through the application of a filter operator. The largest eddies are directly simulated and the smallest eddies are modeled using an appropriate subgrid model. The use of this methodology has been very useful to study fluid mechanics problems, because requires a reasonably small computational cost, and due to its easy implementation.

The main aim of this paper is to present the results of the implementation of the Large Eddy Simulation methodology in a two-dimensional code, based on the finite volume methodology to solve general problems of fluids mechanics and heat transfer. The code has been developed to solve unsteady incompressible flows. The general characteristics of the code are: applicable to solve 2D problems in Cartesian coordinates, uses SIMPLEC –SIMPLE Consistent algorithm to treat the pressure-velocity coupling, uses *Power-law* developed by Patankar (1980) as interpolation scheme and TDMA - *Tri Diagonal Matrix Algorithm* to solve the resulting algebraic equations systems. The first order Euler scheme is used for discretising transient terms and the fully implicit formulation is used for discretising the other terms.

In order to validate the code, the lid-driven square cavity was chosen because of the large amount of data available in the literature and also because of the simplicity of the geometry and boundary conditions. Although the geometric configuration is quite simple, the flow patterns are complexes, allowing rigorous analyses of the code behavior.

2. MATHEMATICAL MODEL

Considering a two-dimensional, incompressible, unsteady and isothermal flow of a Newtonian fluid in Cartesian coordinates, the continuity and momentum equations, Eqs. (1-2), compose the system of equations that governing the lid-driven cavity problem:

$$\frac{\partial u_j}{\partial x_j} = 0 \text{ , with } j = 1,\, 2 \tag{1}$$

$$\frac{\partial u_j}{\partial t} + \frac{\partial u_i u_j}{\partial x_i} = -\frac{1}{\rho}\frac{\partial p}{\partial x_j} + \nu \frac{\partial}{\partial x_i}\left(\frac{\partial u_j}{\partial x_i}\right) + Sc_i \text{ , with } i = 1{,}2 \text{ and } j = 1{,}2 \tag{2}$$

where ρ is the fluid density, ν is the kinematic viscosity, u_j is the velocity vector components and p is the pressure.

3. NUMERICAL SOLUTION METHODOLY

The finite volume methodology is used to solve the flow inside the lid-driven square cavity. Basically, the finite volume method consists of three mains steps. Firstly, the domain is divided in many control volumes, building a discrete computational mesh. In the second step, the governing differential equations are integrated in the control volumes and the resulting equations are discretised resulting in three algebraic equations systems. The algebraic equations systems are solved in the third step. The boundary conditions are also discretised properly.

The interpolation scheme used to treat the advection-diffusive terms is the *Power-law* scheme (Patankar, 1980). The SIMPLEC algorithm is used to solve the pressure-velocity coupling problem and the resulting algebraic equations systems are solved with TDMA – *Tri-Diagonal Matrix Algorithm*. Staggered meshes in relation to pressure are used for the velocity components.

4. TURBULENCE MODELING

4.1. Introduction

The turbulence phenomena has gained special attention in studies associated to computational fluid dynamics. This focus is not justified only by the importance of turbulence for complex engineering problems, but also due to the challenges and obstacles that constitute the numerical analysis of turbulence.

The Navier-Stokes equations are capable to represent both laminar and turbulent flow if the Mach number is less than 15. The order of magnitude of the smallest turbulent eddies, even in these conditions is larger than the mean molecular free path, as discussed in Lessier (1994). For turbulent regime the eddies spectrum is proportional to $Re^{3/4}$. Therefore, for increasing Reynolds numbers, the length of the smallest turbulence eddies decrease, and a refined grid is required to capture these eddies. Many times, expensive computational resources are generally required to obtain those results. This kind of methodology is known as Direct Numerical Simulation (DNS) of turbulence, where all turbulent scales are solved. Because of the prohibitive computational cost, this methodology can be only used for low Reynolds numbers.

In order to solve turbulent flows with the current computational resources, many methodologies has been developed, as the popular RANS/URANS, which are, according to Ferziger and Peric (2002), good models for calculate a few properties of a turbulent flow, such as the average forces on a body, the degree of mixing between two incoming streams of fluid, or the amount of a substance that has reacted. Due to these characteristics and the reasonable computational cost, the RANS/URANS models are very applied to solve engineering problems in industry. In this work, the Large Eddy Simulation (LES) is adopted due to its easy implementation and efficiency to solve complex and unsteady flows, field where the RANS/URANS models do not provide good results. The subgrid scale model proposed by Smagorinsky (1963) is used for modeling the smallest eddies.

4.2. Large Eddy Simulation

Through a filtering process, the largest flow structures are solved directly by the filtered transport equations, while the smallest structures are modeled. The LES and DNS are similar, because both methodologies allow the three-dimensional unsteady solution of the Navier-Stokes equations. LES still demands refined meshes to provide physically consistent results. However, LES makes possible the simulation of turbulent flows for high Reynolds numbers using the existent computational capacity. This characteristic, associated to good results obtained for several complex problems and to its easy implementation, makes LES one of the most promising methodologies to study turbulent flows (Silveira Neto, 2000).

4.3. Filtering Process of the Governing Equations

The filtering process decomposes the variables in a component that represents the largest eddies and a representative subgrid component.

Appling the filter operator in the Eqs. (1) and (2) and using the commutative property between the filter and partial differentiation operator, the new system of equations is:

$$\frac{\partial}{\partial x_j}\left(\overline{\rho u_j}\right)=0 \tag{3}$$

$$\frac{\partial}{\partial t}\left(\overline{u_j}\right)+\frac{\partial \overline{u_i u_j}}{\partial x_i}=-\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_j}+\nu\frac{\partial}{\partial x_j}\left(\frac{\partial \overline{u_j}}{\partial x_i}\right) \tag{4}$$

Splitting the velocity components as

$$u_j=\overline{u}_j+u'_j \tag{5}$$

and according to the properties of the filter operator, the non-linear convective term can be written as:

$$\overline{u_i u_j}=\overline{(\overline{u}_i+u'_i)(\overline{u}_j+u')}=\overline{\overline{u}_i\overline{u}_j}+\overline{\overline{u}_i u'_j}+\overline{\overline{u}_j u'_i}+\overline{u'_i u'_j} \tag{6}$$

Defining the subgrid Reynolds stress tensor, the cross stress tensor, and the Leonard tensor as:

$$\tau_{ij}=\overline{u'_i u'_j} \qquad \textit{(Subgrid Reynolds Stress Tensor)} \tag{7}$$

$$C_{ij}=\overline{u'_i\overline{u}_j}+\overline{\overline{u}_i u'_j} \qquad \textit{(Cross-stress tensor)} \tag{8}$$

$$L_{ij}=\overline{\overline{u}_i\overline{u}_j}-\overline{u}_i\overline{u}_j \qquad \textit{(Leonard tensor)} \tag{9}$$

one can rewrite Equation (6) as:

$$\overline{u_i u_j}=\overline{u}_i\overline{u}_j+L_{ij}+C_{ij}+\tau_{ij} \tag{10}$$

Using these new variables, the governing equations can be written as fallow:

$$\frac{\partial \overline{u}_i}{\partial x_i}=0 \tag{11}$$

$$\frac{\partial \overline{u}_i}{\partial t}+\frac{\partial}{\partial x_j}\left(\overline{u}_i\overline{u}_j\right)=-\frac{1}{\rho_o}\frac{\partial \overline{p}}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\nu\left(\frac{\partial \overline{u}_i}{\partial x_j}+\frac{\partial \overline{u}_j}{\partial x_i}\right)-\left(\tau_{ij}+C_{ij}+L_{ij}\right)\right] \tag{12}$$

This is a system composed by four equations and twenty and two unknowns, the four filtered primitive unknowns ($\overline{u}_i$ e $\overline{p}$) and the eighteen new tensors (τ_{ij}, C_{ij} e L_{ij}). This problems can be solved by using turbulent models.

In the next sections, it is presented the subgrid model to calculate the eddy viscosity, emphasizing the model proposed by Smagorinsky (1963).

4.4. Turbulence subgrid modeling

In the present work the cross-stress and the Leonard tensors were negleted. This hypothesis is based on the works of Shaanan *et al*. (1975) and Silveira-Neto *et al*. (1993). Shaanan *et al*. (1975) concluded that, when a second order scheme or less is used for dicretization of the advective term, the cross-stress and Leonard tensors can be disregarded. On the other hand, when high order schemes or spectral methods are used, these tensors must be considered. However, Silveira-Neto *et al.* (1993), have show that these two tensors can be disregarded even for third order schemes.

Using the Boussinesq hypothesis (Boussinesq, 1877), which has considered the subgrid Reynolds stress tensor as a function of the strain rate tensor of the filtered velocity field and the turbulent kinetic energy (k), as follows:

$$\tau_{ij} = \frac{2}{3} k \delta_{ij} - \nu_t \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \tag{13}$$

where ν_t is the eddy viscosity, the Eqs (11) and (12) can be rewritten as:.

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0 \tag{14}$$

$$\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial}{\partial x_j}\left(\bar{u}_i \bar{u}_j\right) = -\frac{1}{\rho}\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\nu \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) + \nu_t \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \frac{2}{3} k \delta_{ij} \right] \tag{15}$$

If the kinetic energy (k) is not calculated, the modified pressure (p^*) incorporates the effects from the variations of the subgrid turbulent kinetic energy in the flowfield. In this case, the final governing equations are:

$$\frac{\partial \bar{u}_i}{\partial x_i} = 0 \tag{16}$$

$$\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial}{\partial x_j}\left(\bar{u}_i \bar{u}_j\right) = -\frac{1}{\rho}\frac{\partial \overline{p^*}}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\left(\nu + \nu_t\right) \left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) \right] \tag{17}$$

which is the closed system of equations to be solved. In order to solve this system the eddy viscosity (ν_t) must be modeled. In the next section is presented the Smagorinsky (1963) model adopted in this work.

4.5. Smagorinsky Subgrid Model

The Smagorinsky model is the most popular for obtaining the eddy viscosity, ν_t, and has been developed to simulated atmospheric flows. This model is based on the assumption of local equilibrium for the smallest eddies, where the production of the subgrid turbulent stress ($\wp$) is considered equal to the dissipation rate (ε):

$$\wp = \varepsilon \tag{18}$$

where the production of subgrid turbulent stress can be expressed as a function of the strain rate of the filtered field and the dissipation can be expressed as a function of the velocity and the subgrid characteristic length, as:

$$\wp = -\overline{u_i' u_j'} S_{ij} = 2 \nu_t S_{ij} S_{ij} \tag{19}$$

$$\varepsilon = -c_1 \overline{(u_i' u_j')}^{3/2} / \ell \tag{20}$$

The subgrid eddy viscosity, ν_t, is considered to be proportional to these two scales, as follows:

$$\nu_t = c_1 \ell \overline{(u_i' u_j')} \tag{21}$$

$$\nu_t = (C_S \ell)^2 |\overline{S}| \tag{22}$$

where C_S is Smagorinsky constant, ℓ is the scale length, calculated as a function of the discretised mesh, and $|\overline{S}|$ is the absolute value of the strain rate tensor, defined as:

$$|\overline{S}| = \left(2\overline{S}_{ij}\overline{S}_{ij}\right)^{1/2} \tag{23}$$

where S_{ij} is given by:

$$S_{ij} = \frac{1}{2}\left(\frac{\partial \overline{u}_i}{\partial x_j} + \frac{\partial \overline{u}_j}{\partial x_i}\right) \tag{24}$$

The scale lenght ℓ can be calculate by:

$$\ell = (\Delta x \cdot \Delta y)^{1/2} \tag{25}$$

An effective viscosity, ν_{ef} , can be defined as:

$$\nu_{ef} = \nu_t + \nu \tag{26}$$

The Smagorinky constant (C_S) has a value of order 0.1. For isotropic and homogenous turbulence, Lilly (1967) has analytically determined that C_S = 0.18. Actually, the value of this constant depends on the characteristics of the computational code and on the grid used in the simulation, and should be adjusted for each problem. One of the inconsistencies of the Smagorinsky model is the overestimation of the eddy viscosity near the walls, specially in the boundary layer. For many cases the use of a function to reduce the eddy viscosity in that region can be necessary.

5. CONFIGURATION OF THE NUMERICAL SIMULATIONS

The study of the lid-driven square cavity problem is a standard test used to validate numerical codes. This problem is characterized by simple geometry and boundary conditions, but presents complex flow patterns. Therefore, it is a very useful problem to analyze the quality of the computational code. Figure 1 shows a scheme of the lid-driven square cavity as well as the used boundary conditions.

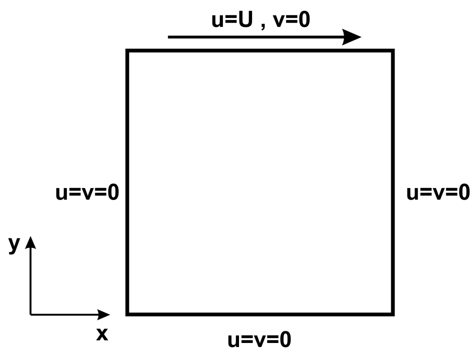


Figure 1. Scheme of the lid-driven square cavity and boundary conditions.

The problems was simulated for Reynolds numbers varying from transition to the turbulent regimes, with the Reynolds number based on the height of the cavity and on the prescribed velocity U. Simulations for low Reynolds numbers were not presented in this work once the code has already been validated for the lid-driven square cavity using DNS, with excellent results, as can be seen in Rodrigues and Gasche (2008).

A maximum mass residue over the entire domain less than 10^{-10} was used as a convergence criteria.

6. NUMERICAL RESULTS

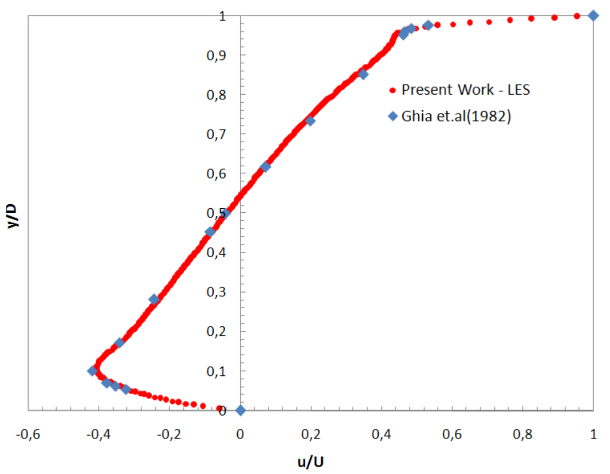
For evaluating the results of the flow in the lid-driven cavity, it is performed an analysis of the dimensionless velocity profiles u/U and v/U. The problem was simulated for the following Reynolds numbers: $3.2x10^3$, $5x10^3$, $7.5x10^3$ and 10^4. The Smagorinsky constant adjusted for each simulation case is presented in Tab. 1.

Table 1. Smagorinsky constant adjusted for each simulation.

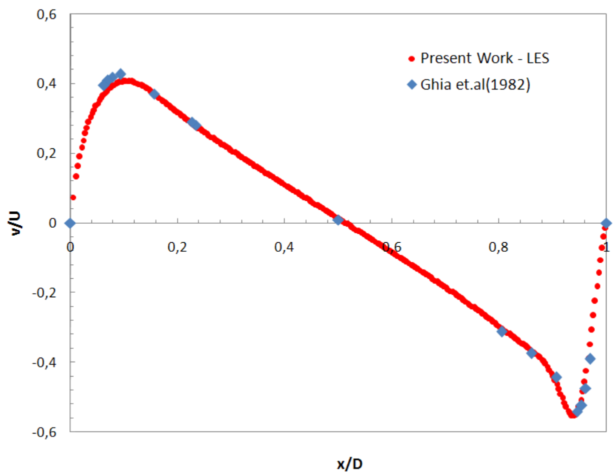
Re	Cs
3200	0.18
5000	0.18
7500	0.30
10000	0.30

The results were confronted with the data obtained by Ghia *et al.* (1982) who used a mesh configuration of 257x257 volumes. The same configuration for the computational mesh was adopted in this work for purposes of comparison.

Figures 2 to 5 present the dimensionless velocity profiles obtained for Reynolds numbers of $3.2x10^3$, $5x10^3$, $7.5x10^3$ and 10^4, respectively.

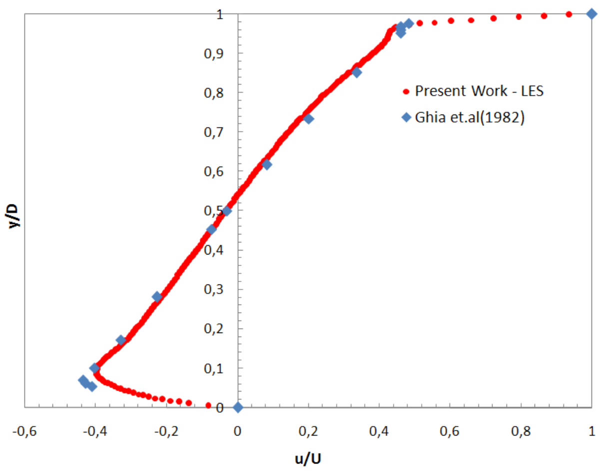


(a)

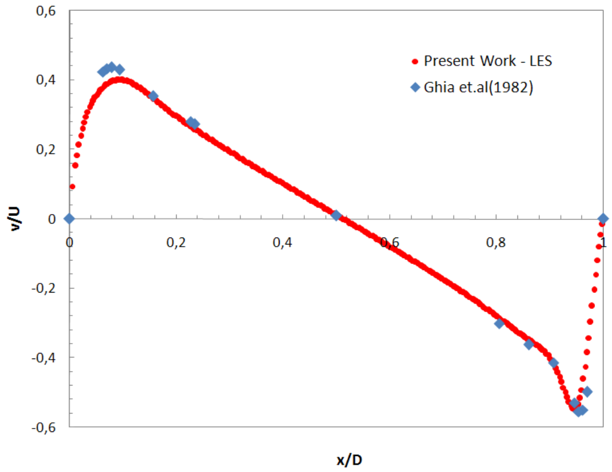


(b)

Figure 2. Dimensionless velocity profiles for Re = 3200.

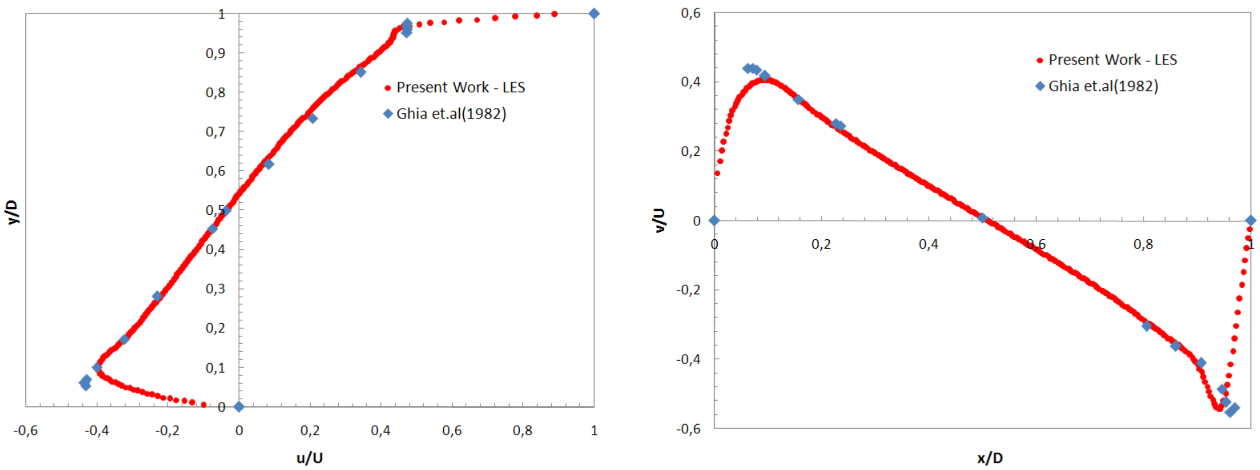


(a)



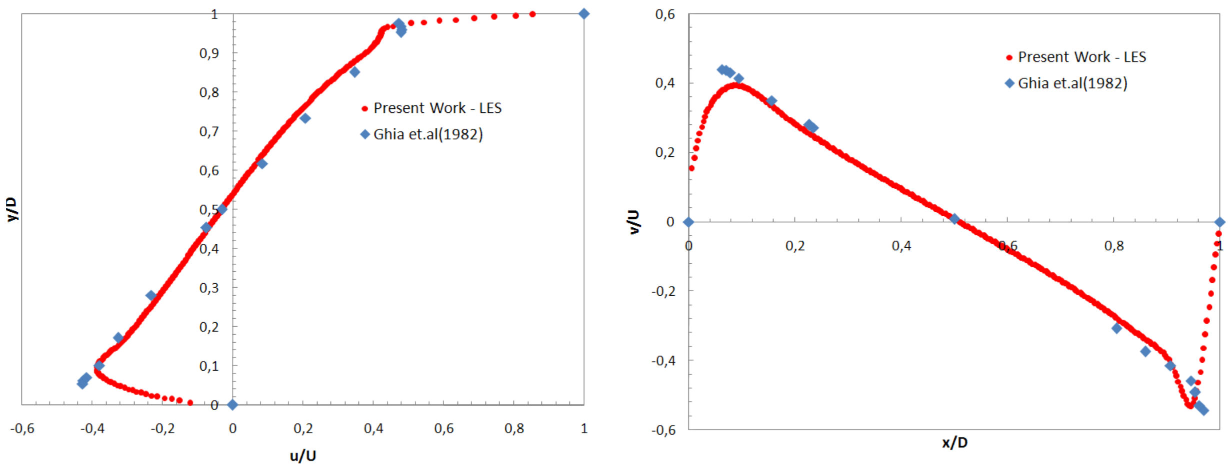
(b)

Figure 3. Dimensionless velocity profiles for Re = 5000.



(a) (b)

Figure 4. Dimensionless velocity profiles for Re = 7500.

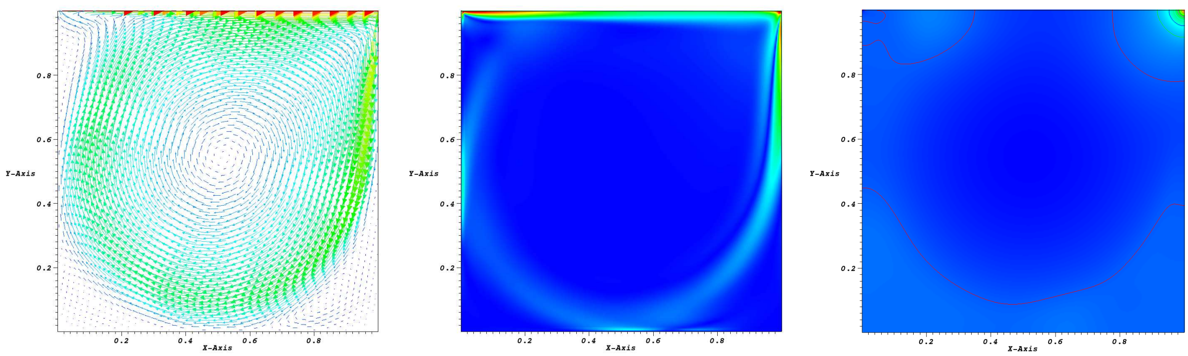


(a) (b)

Figure 5. Dimensionless velocity profiles for Re = 10000.

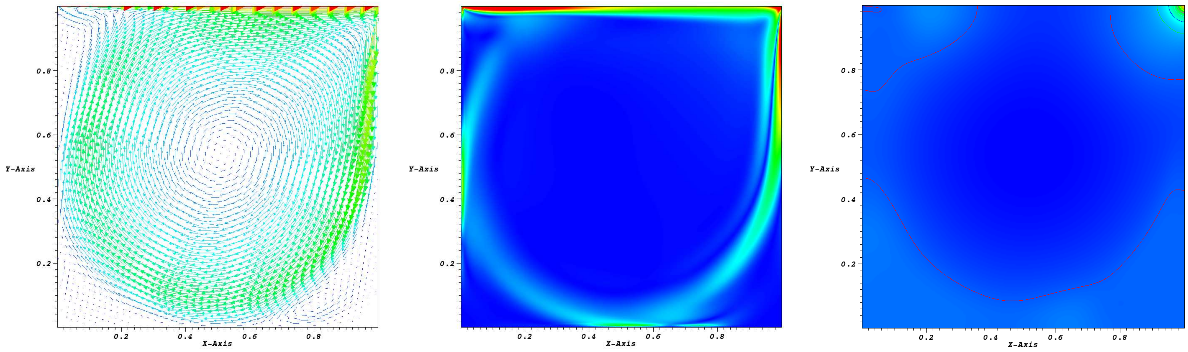
These figures show that the velocity profiles have good agreement with data from Guia *et al.* (1982), showing good consistence of the implemented turbulence model. The most significant deviations is founded in the region of large gradients, and for the highest Reynolds numbers. This deviation is probably caused by the interpolation function used to treat the advective-difusive terms (*Power-law*), which is quite efficient, but presents significant numerical diffusion. In addition , the Smagorinsky model, overestimate the eddy viscosity near the walls, introducing excessive diffusivity in these regions, contributing to the deviation of the velocity profiles.

Figures 6 to 9, present maps for velocity vectors, eddy viscosity and pressure obtained for the same Reynolds numbers.



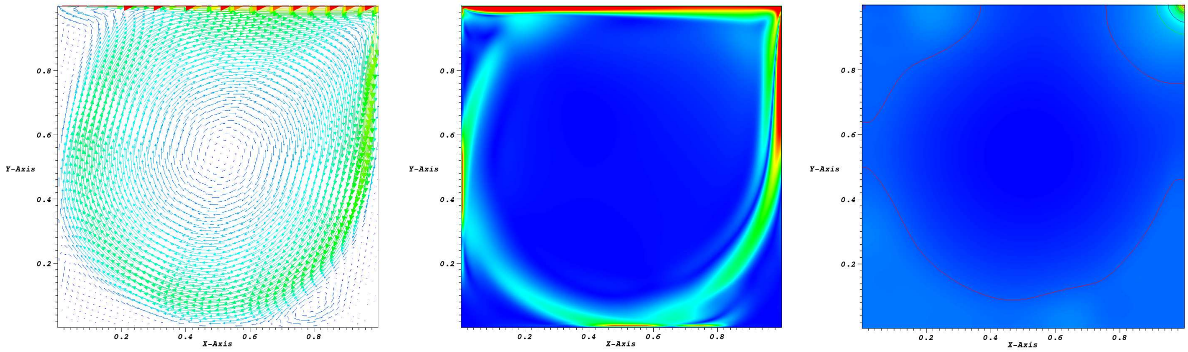
(a) (b) (c)

Figure 6. Maps of (a) velocity vector, (b) eddy viscosity and (c) pressure field for Re = 3200.



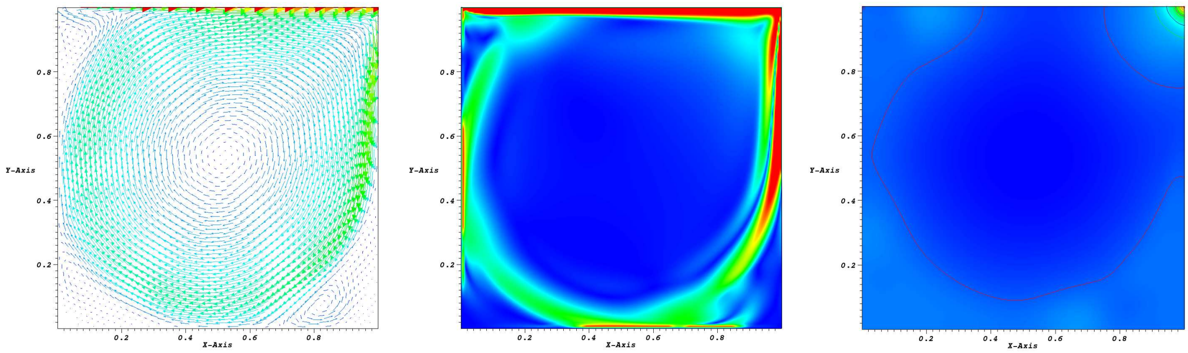
(a) (b) (c)

Figure 7. Maps of (a) velocity vector, (b) eddy viscosity and (c) pressure field for Re = 5000.



(a) (b) (c)

Figure 8. Maps of (a) velocity vector, (b) eddy viscosity and (c) pressure field for Re = 7500.



(a) (b) (c)

Figure 9. Maps of (a) velocity vector, (b) eddy viscosity and (c) pressure field for Re = 10000.

Observing the maps of velocity vector one can noted the large central vortex characteristic of this kind of flow, and the evolution of the secondary recirculation's in both inferior and superior left corners.

As can be observed on the maps of eddy viscosity, the values of the eddy viscosity increase for increasing Reynolds numbers, once it is proportional to the strain rate of the flow.

7. CONCLUSIONS

The development of the present work focuses on the implementation of the Large Eddy Simulation methodology for turbulence modeling, using the subgrid model proposed by Smagorinsky (1963) in an existent computational code constructed for solving general flow problems using the finite volume method. The code with the turbulence model was validated for the lid-driven square cavity flow problem.

The problem was simulated for Reynolds numbers varying from the transitional to turbulent regimes. The results presented excellent agreement with the data from Ghia *et.al* (1982) for Reynolds numbers until $5x10^3$, presenting larger deviation for $7.5x10^3$. This deviation is attributed partially to the interpolation scheme for the advective-difusive terms (*Power-law*), and to the subgrid model, which overestimate the eddy viscosity near the walls.

The obtained results showed that the computational code works well for transitional and turbulent flows in lid-diven cavities using the turbulence model implemented, and can be tested in future studies of other problems.

8.ACKNOWLEDGEMENTS

The authors acknowledge the support given by FAPESP – São Paulo State Research Foundation.

9.REFERENCES

Boussinesq, J., 1877, “Théorie de l’Écoulement Tourbillant”, Mem. Présentés par Divers Savants Acad. Sci. Inst. *Fr.*, vol. 23, p.46-50.

Ferziger, J. H.; Peric, M., 2002 , “Computational Methods for Fluid Dynamics”, 3rd Ed. Springer.

Ghia, V., Ghia. K. N., Shur, C. T., 1982 , “High-Re Solutions for Incompressible Flow Using the Navier-Stokes Equations and a Multigrid Method”. J. of Computational Physics, vol.48, p.387-411.

Lesier, M., 1994, “La Turbulence”, Presses Universitaires de Grenoble, France.

Lilly, D. K., 1967, “The Representation of Small-Scale Turbulence in Numerical Experiments”, Proc. IBM Sci. Comput. Symp. Environ. Sci., IBM Data Process. Div., White Plains, p. 195-210.

Patankar, S.V., 1980, “Numerical Heat Transfer and Fluid Flow”, Ed. Hemisphere.

Rodrigues, T. T. ; Gasche, J. L., 2008, “Numerical Study of Unsteady Laminar Flow in Lid-driven Square Cavities” (in Portuguese). In: *DINCON*-Brazilian Conference on Dynamics, Controls and Applications, 2008, Presidente Prudente-SP. DINCON Annals 2008.

Shaanan, S., Ferzinger, J. H. e Reynolds, W. C., 1975, “Numerical Simulation of Turbulence in Presence of Shear”, Rep. TF-6, Dept. Mechanical Engineering, Stanford University.

Silveira-Neto, A., 2000, “Applied Turbulence in Fluids” (in Portuguese), Postscript, Universidade Federal de Uberlândia, Uberlândia – MG, Brasil.

Silveira Neto, A., Grand, D., Métais, O., Lesieur, M., , 1993, “A Numerical Investigation of the Coherent Vortices in Turbulence behind a Backward-Facing Step”, J. Fluid Mechanics, vol.256, p.1-25.

Smagorinsky, J., 1963, “General Circulation Experiments with the Primitive Equations I. The Basic Experiment”, *Mon. Weather Rev*., vol. 91, p.99-164.

10. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.