NUMERICAL ANALYSIS OF THREE-DIMENSIONAL TURBULENT FLOW BY FINITE ELEMENT METHOD AND LARGE EDDY SIMULATION

Adriane Prisco Petry
Department of Mechanical Engineering
Federal University of Rio Grande do Sul
Sarmento Leite, 425, CEP:90050-170, Porto Alegre, RS, Brazil
adrianep@mecanica.ufrgs.br

Armando Miguel Awruch
Civil Engineering Graduate Program
Federal University of Rio Grande do Sul
Osvaldo Aranha, 99, CEP:90035-190, Porto Alegre, RS, Brazil
awruch@adufrgs.ufrgs.br

Abstract: Formulation, implementation and applications of a numerical algorithm to simulate turbulent, incompressible, isothermal flows are the main objectives of this work. The transient three-dimensional flow is analyzed using an explicit Taylor-Galerkin scheme and the finite element method with hexahedrical eight-node element. The scheme adopted for turbulence treatment is Large Eddy Simulation. For sub-grid scales two models where implemented, the classical Smagorinsky´s model and the dynamic eddy viscosity model. For the process of second filtration, which is necessary in the dynamic model, a new method was developed based on independent finite elements that involve each node in the original mesh. The implemented scheme is efficient and good results with low additional computational cost were obtained. Results for classical problems where presented, that demonstrate the system validation. Comments about the scheme applicability for flows with high Reynolds number are presented in the last part.

Keywords: Large Eddy Simulation, Finite Element Method, Dynamic Model, Turbulence, Computational Fluid Dynamics

1. Introduction

Flow analysis is an important subject for several engineering fields, as well as in other areas of science and technology. Many problems are characterized by turbulent flows, even in engineering, as in meteorology, medicine or in atmospheric dispersion of pollutants. There are areas of interest, such as the aerodynamic project and optimization (Reuther et al, 1999), that require a precise flow determination, these are applications where the improvement of the methods of turbulent flow analysis is essential.

Turbulent flows are usually characterized by high Reynolds numbers, a coherent behavior at large scales level and random behavior at small scales, they are also diffusive, three-dimensional and transient (Tenekes and Lumley, 1972). Another important characteristic of turbulent flows is that multiple scales are involved (Silveira Neto, 2002). However even small scales are usually greater than the scales of molecular movement (Hinze, 1975), than turbulence may be described as a continuous phenomenon.

Using the conservation equations of mass, energy and momentum, as the mathematical model for flow analysis, a complex system of partial differential equations is obtained Computational Fluid Dynamics, is an important methodology to solve the problem. Different numerical methods are used to carried out these simulations. The Finite Element Method (Hughes, 1987; Reddy and Gartling, 1994) is an efficient technique that presents proper characteristic for the analysis of problems with complex geometry (Brazil Junior, 2002). This methodology was adopted in the present work.

The conservation equations of fluid mechanics consists a complete mathematical model, which is able to describe turbulent fluid flows, however the required discretization in space and time for simulation of all involved scales directly (Direct Simulation), becomes impracticable this type of analysis, for most of practical problems. This is a consequence of large number of equations to be solved (Grötsbach, 1987), leading to very large processing times, even for the most advanced computers (Kim and Menon, 1999). As consequence of the impossibility to apply Direct Simulation for a large range of problems, it is necessary to use alternative methodologies, such as the classic modeling based on the solution of the Reynolds Average Equations (Hinze, 1975) and the Large Eddy Simulation (Ferziger 1993, Rogallo and Moin 1984, Lesieur et al, 1995).

In the Large Eddy Simulation technique, the conservation equations are solved for large flow scales directly and models are used to represent the effect of the subgrid scales. These models have the same purpose of the conventional turbulence models, however it is possible to apply simple ones, once they must consider only the effect of small scales, moreover, subgrid models have minor geometry dependence, therefore the small scales are of more universal nature than the total turbulence.

This work presents the formulation, implementation and application of a numerical algorithm for three-dimensional, turbulent flow analysis, more detailed in Petry, 2002. The methodology is based on the Finite Element

Method and Large Eddy Simulation. A computational code to simulate transient, quasi-incompressible, three-dimensional flows, was developed using an explicit Taylor-Galerkin scheme, with eight-node hexahedrical element. Two different subgrid models were implemented, the Smagorinsky´s model (Smagorinsky, 1963) and the dynamic model (Germano et al., 1991; Lilly, 1992). The dynamic model implementation implies in a second filter operation. It was developed a new methodology for this process, called Second Filter by Independent Finite Elements. Simulations of different flows over a backward-facing step are presented in this work. These simulations confirm the validity of the implemented scheme, however they also demonstrate that improvements are necessary to overcome difficulties presented to simulate large computational problems. It was verified, in these simulations, that a very small integration time step is required for convergence and consequently the system becomes very expensive in processing time.

2. Mathematical and Numerical Aspects

2.1 Governing Equations

The flows analyzed in this work are incompressible, then it is necessary to adopt some scheme to overcome difficulties in numerical analysis of these kind of flows (Awruch and Petry, 1997, Reddy and Gartling, 1994). Usual formulation for incompressible flow is on based the assumption of a constant value for the density and, from this hypothesis, it is deduced that the speed of sound in the flow field is infinite (Schlichting, 1968), however in real flows the propagation of sound always occurs with a finite speed. At the present work, the equations for a quasi-incompressible flow are adopted (Kawahara and Hirano, 1983), which assume constant density, but a finite value for the speed of sound. With this consideration the equation of mass conservation contains the time derivative of pressure, preventing zeros in the main diagonal line of the mass matrix in the finite element formulation.

From the equations of conservation of mass, energy and momentum for three-dimensional, transient, isothermal, quasi-incompreessible viscous flows, of a Newtonian fluid (White, 1974, Kawahara and Hirano, 1983), the equations for Large Eddy Simulation are deduced (Petry 2002).

In a Large Eddy Simulation (Findikakis and Street, 1982) each field variable is decomposed into large scale field (identified by the over-bar) and subgrid scale field (identified by the apostrophe):

$$v_i = \overline{v_i} + v_i' \qquad p = \overline{p} + p' \qquad \rho = \overline{\rho} + \rho' \tag{1}$$

Since density is constant, then $\rho'=0$.

Proceeding the filtering process of the equations of conservation of mass and momentum of the isothermal, viscous, quasi-incompressible, three-dimensional and transient flow, the governing equations are given by:

$$\frac{\partial \overline{p}}{\partial t} + C^2 \frac{\partial}{\partial x_j}\left(\rho \overline{v}_j\right) = 0 \tag{2}$$

$$\begin{aligned} &\frac{\partial}{\partial t}\left(\rho \overline{v}_i\right) + \frac{\partial}{\partial x_j}\left(\rho \overline{v}_i \overline{v}_j\right) + \frac{\partial \overline{p}}{\partial x_j}\delta_{ij} - \frac{\partial}{\partial x_j}\left\{ \nu\left(\frac{\partial}{\partial x_j}(\rho \overline{v}_i) + \frac{\partial}{\partial x_i}(\rho \overline{v}_j)\right) + \frac{\lambda}{\rho}\left(\frac{\partial}{\partial x_k}(\rho \overline{v}_k)\right)\delta_{ij} \right\} \\ &\qquad + \frac{\partial}{\partial x_j}\left\{\rho\left(L_{ij} + C_{ij} + \overline{v_i' v'_j}\right)\right\} - f_i = 0 \qquad (j = 1,2,3)\ \text{in}\ \Omega \end{aligned} \tag{3}$$

With the following boundary conditions:

$$v_i = \hat{v}_i \qquad \text{in } \Gamma_v \tag{4}$$

$$\left\{\left[-\overline{p} + \frac{\lambda}{\rho}\frac{\partial}{\partial x_k}\left(\rho \overline{v_k}\right)\right]\delta_{ij} + \nu\left[\frac{\partial}{\partial x_j}\left(\rho \overline{v_i}\right) + \frac{\partial}{\partial x_i}\left(\rho \overline{v_j}\right)\right]\right\} n_j = t_i \qquad \text{in } \Gamma t \tag{5}$$

and the corresponding initial conditions:

$$v_i = \hat{v}_{i0} \qquad \text{in } t = 0, \Omega \tag{6}$$

$$p = \hat{p}_0 \qquad \text{in } t = 0, \Omega \tag{7}$$

Where:
ρ - density
x_i – i direction coordinate

δ_{ij} - Kroenecker delta
n_j – cossin director of normal vector at the boundary
t_i – prescribed values of surface forces at the boundary
λ - volumetric viscosity
C- sound propagation speed
ν -cinematic viscosity.

$\overline{v}_i$ -large scale velocity component in x_i direction

$\overline{p}$ - large scale pressure component

v'_i - subgrid scale velocity component in x_i direction

$L_{ij} = \overline{\overline{v}_i\,\overline{v}_j} - \overline{v}_i\,\overline{v}_j$, Leonard´s terms

$Cij = \overline{\overline{v}_i\,v_j'} + \overline{v_i'\,\overline{v}_j}$, Cross terms

$\overline{v_i'\,v_j'}$ subgrid Reynolds stresses

$\hat{v}_i$ - prescribed values of velocity at the boundary

The L_{ij} and C_{ij} terms can be neglected (Findikakis and Street, 1982). Previous studies (Petry and Awruch, 1997b) confirm that the consideration of these terms does not affect the results significantly and increase around 20% the processing time.

Equations (2) and (3), neglecting the Leonard´s and crossed terms, with the boundary and initial conditions given by equations (4), (5), (6) and (7), are the governing equations of the turbulent, isothermal, quasi-incompressible flow, of a Newtonian fluid and, with te subgrid scales models equations, consist the system to be solved.

2.2 Subgrid Scale Models

The two implemented models are based on eddy viscosity concept, using the hypothesis of Bousinesq, the subgrid Reynolds stresses are given by:

$$-\overline{v_i'\,v_j'} = \nu_t \left(\frac{\partial \overline{v}_i}{\partial x_j} + \frac{\partial \overline{v}_j}{\partial x_i} \right) \tag{8}$$

Where ν_t is the eddy viscosity.

This is the original equation of Bousssinesq. Usually for incompressíble flows the equation (8) is modified, introducing a term with subgrid kinetic energy to make the model compatible with the usual equation of mass conservation for incompressible flows (Hinze, 1975). However in this work the continuity equation is the modified one, for quasi-incompressible flows, therefore, equation (8) is valid.

2.3 Smagorinsky´s Model

The model of Smagorinsky (1963) has been traditionally used to represent the effect of the subgrid scales in Large Eddy Simulation (Findikakis and Street, 1983; Lesieur et al, 1995). It is a eddy viscosity model in which the subgrid Reynolds stresses are given by the equation (8) and eddy viscosity is defined as:

$$\nu_t = C_S^2\, \overline{\Delta}^2 \left| \overline{S} \right| \tag{9}$$

Where C_S is the Smagorinsky´s constant, with values from 0.1 to 0.22, and the other terms are given by:

$$\left| \overline{S} \right| = \sqrt{2 \overline{S}_{ij} \overline{S}_{ij}} \tag{10}$$

$$\overline{S}_{ij} = \frac{1}{2} \left(\frac{\partial \overline{v}_i}{\partial x_j} + \frac{\partial \overline{v}_j}{\partial x_i} \right) \tag{11}$$

$$\overline{\Delta} = \sqrt[3]{\prod_{i=1}^{3} \Delta x_i} \tag{12}$$

2.4 Eddy Viscosity Dynamic Model

The dynamic model was first proposed by Germano et al., 1991, and modified by Lilly, 1992. The subgrid Reynolds stresses are also obtained with equation (8), however the eddy viscosity is defined by:

$$\nu_t = C(\mathbf{x},t)\overline{\Delta}^2 \left| \overline{S} \right| \tag{13}$$

The dynamic coefficient is calculated as a function of the local flow characteristics, using a double filtering process. The calculation of C(x, t) is based on information of the small scales solved by the mesh, and is defined as:

$$C(\mathbf{x},t) = -\frac{1}{2}\frac{L_{ij} M_{ij}}{M_{ij} M_{ij}} \tag{14}$$

Where Lij e Mij are given by:

$$L_{ij} = \left\langle \overline{v}_i \overline{v}_j \right\rangle - \left\langle \overline{v}_i \right\rangle \left\langle \overline{v}_j \right\rangle \qquad M_{ij} = \left\langle \overline{\Delta} \right\rangle^2 \left| \left\langle \overline{S}_{ij} \right\rangle \right| \left\langle \overline{S}_{ij} \right\rangle - \left\langle \overline{\Delta}^2 \left| \overline{S}_{ij} \right| \overline{S}_{ij} \right\rangle \tag{15}$$

$$\left\langle \overline{S}_{ij} \right\rangle = \frac{1}{2}\left(\frac{\partial \left\langle \overline{v}_i \right\rangle}{\partial x_j} + \frac{\partial \left\langle \overline{v}_j \right\rangle}{\partial x_i} \right) \qquad \left| \left\langle \overline{S} \right\rangle \right| = \sqrt{2 \left\langle \overline{S}_{ij} \right\rangle \left\langle \overline{S}_{ij} \right\rangle} \tag{16}$$

$\left\langle \overline{\Delta} \right\rangle$ - characteristic length of the second filter, with $\left\langle \overline{\Delta} \right\rangle > \overline{\Delta}$.

In the above equations, the bar indicates the first process of filtering (filter at mesh level) and the symbol $\langle \ \rangle$ indicates the second filtering process (test filter).

For the solution of the system of equations, Finite Element Method is employed. To get the system of algebraic equations, time derivatives are expanded in Taylor series, including the second order terms and for the space discretization the classic Galerkin method is applied (Reddy and Gartling, 1994). To save processing time, analytical expressions for the elements matrices are used and the hexaedrical isoparametric element (Burbridge e Awruch,2000). This scheme is known as Taylor-Galerkin, (Donea, 1984), and was used by Azevedo, 1999, for the simulation of three-dimensional laminar flows with fluid-structure interaction. The scheme is explicit and conditionally stable and the integration time step has the following restriction:

$$\Delta t \leq \frac{\Delta x_i (\min)}{C + V} \tag{17}$$

where $\Delta x_i (\min)$ is the minimum dimension of the mesh elements, C is the speed of sound and V is the reference velocity. Details of the numerical methodology can be found in Petry, 2002 and Azevedo, 1999.

2.5 The Second Filter: Proposed Methodology

The equation (14) defines the dynamic coefficient, C(x, t). This coefficient depends on the use of two filters of different characteristic lengths. The first filter, at the mesh level, has characteristic length related to element dimension. For the second filtering process (test filter) the characteristic length must be greater than the length of the first filter. Based on these two scale levels, the dynamic model uses information of the smallest resolved scales (situated between the two filters) to calculate the dynamic coefficient employing equations (14), (15) and (16).

For the process of second filtering many proposals have been presented. Oshima, et al., 1996, formulate the second filtering operation in a Finite Element code using expansions in Taylor series. Padilla and Silveira Neto, 2000, present and compare different methodologies in the context of Finite Volumes.

The new methodology used in this work was presented in Petry, 2002. The Second Filter by Independent Finite Elements uses techniques common to finite elements, which are: the definition of elements by conectivities; the use of two systems of coordinates (global (x_1,x_2,x_3) and natural (ξ, η, ζ)); the transformations of coordinates and elements interpolation functions. The scheme consists in create one super-element around each node of the mesh and, with the usual shape functions, a linear interpolation of the values calculated in the super-element nodes is performed to get filtered values in the node of interest.

The first stage is included in the pre-processing phase, and consists of generating a list of conectivities of the independent super-elements for each node.. In this scheme the test filter dimension is not prescribed, but it is possible to include some restriction with respect to such dimension. In Figure 1 an example of an independent super-element is presented.

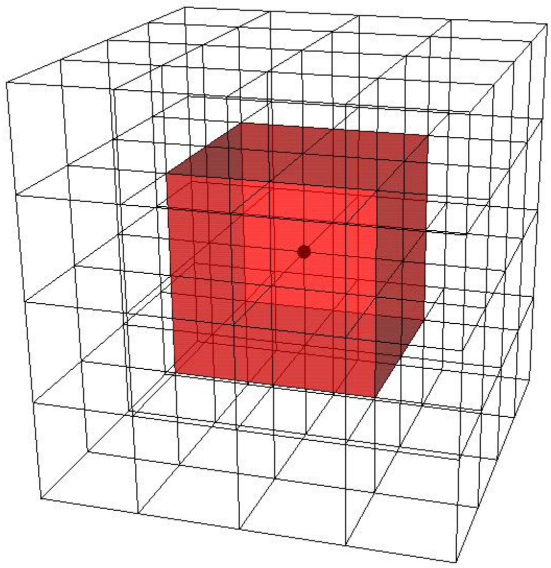

Figure 1 – Independent Super-Element created around a node, on a three-dimensional mesh, to be used in the second filtering process.

The second stage, also included in the pre-processing phase, is to evaluate the natural coordinates (ξ_I, η_I, ζ_I) of node I in the interior of its independent super-element. Global coordinates of a point I inside of an element can be calculated by the following transformation of coordinates:

$$(x_i)_I = \sum_{\alpha=1}^{8} \phi_\alpha(\xi_I, \eta_I, \zeta_I)\, x_{i\alpha} \tag{18}$$

where

$(x_i)_I$ - coordinate of any point I inside an independent super-element, (i=1,2,3)

$x_{i\alpha}$ - coordinate of node α, (α =1,2,3,4,5,6,7,8), (i=1,2,3)

$\phi_\alpha(\xi_I, \eta_I, \zeta_I)$ - interpolation function of node α, evaluated at point I, with natural coordinates (ξ_I, η_I, ζ_I)

This is the usual transformation used in finite elements. However, it is necessary to solve the inverse problem, to get the natural coordinates of a point inside the element (ξ_I, η_I, ζ_I), from its global coordinates, $(x_1,x_2,x_3)_I$, and the global coordinates of the eight nodes of the super-element, $(x_1,x_2,x_3)_\alpha$. In the analysis of this problem a non linear system of three equations is derived and an algorithm using an iterative process of solution was implemented to solve the system.

The two first stages were developed in the pre-processing phase, and they do not represent significant additional cost in the flow analysis system. The stage included in the main algorithm evaluates the terms with second at each node inside of the independent super-element associate to this node, in the following form:

$$\langle \bar{v}_i \rangle_I = \sum_{\alpha=1}^{8} \phi_\alpha(\xi_I, \eta_I, \zeta_I)\, \bar{v}_{i\alpha} \tag{19}$$

$$\langle \bar{v}_i \bar{v}_j \rangle_I = \sum_{\alpha=1}^{8} \phi_\alpha(\xi_I, \eta_I, \zeta_I)\, \bar{v}_i \bar{v}_{j\,i\alpha} \tag{20}$$

$$\left\langle \bar{\Delta}^2 \left|\bar{S}_{ij}\right| \bar{S}_{ij} \right\rangle_I = \sum_{\alpha=1}^{8} \phi_\alpha(\xi_I, \eta_I, \zeta_I) \left(\bar{\Delta}^2 \left|\bar{S}_{ij}\right| \bar{S}_{ij} \right)_\alpha \tag{21}$$

Using these equations, the values of neighboring nodes are weighed, through a linear interpolation, obtaining the filtered value in accordance with the distance to the node I. Characteristic dimension of test filter is calculated in the same way of the first filter, given by equation (12), considering the dimensions of the independent super-elements.

The eddy viscosity is evaluated at the element level and the values of the dynamic coefficient are calculated for each node of the mesh, so the coefficient used for each element is the average of the of C(x, t) of the nodes to each element in the original mesh. This procedure is in accordance with methods of other authors (Oshima et al, 1996; Zang et al, 1993, Breuer and Rodi, 1994), that uses averages of the dynamic coefficient to prevent abrupt variations in space and time, source of instabilities in the solution. Another technique, cited by Lilly, 1992, consists of proceeding averages from the terms Mij and Lij before the calculation of C(x, t), stabilizing the problem and preventing zeros in the denominator.

In this work, a limit for negative values of the eddy viscosity was adopted, which is expressed in equation (22). The same limit was used by Zang et al, 1993.

$$\nu + \nu_t \geq 0 \tag{22}$$

Another verification adopted here is when the denominator of the expression for C(x, t) is zero, it is assumed C(x, t)=0, in the corresponding node.

This scheme increases total processing time between 9 and 18%, in relation to the use of the Smagorinsky´s model , for the problems analyzed in this work.

3. Numerical Example

Flow simulations of a two-dimensional backward-facing step with low Reynolds numbers, were initially performed in order to validate the code. The results are obtained using the codes with Smagorinsky's model and with dynamic model, they were compared with experimental data (Armaly et al., 1983) and other numerical simulations (Silveira Neto et al., 1993; Kaiktsis et al, 1991).

The problem domain is presented in Figure 2. For the two-dimensional case, there is only one element in x_3 direction, and the components of the velocity in this direction are equal to zero (v_3=0) over the whole flow field. The dimensions are similar to the experimental work of Armaly et al, 1983.

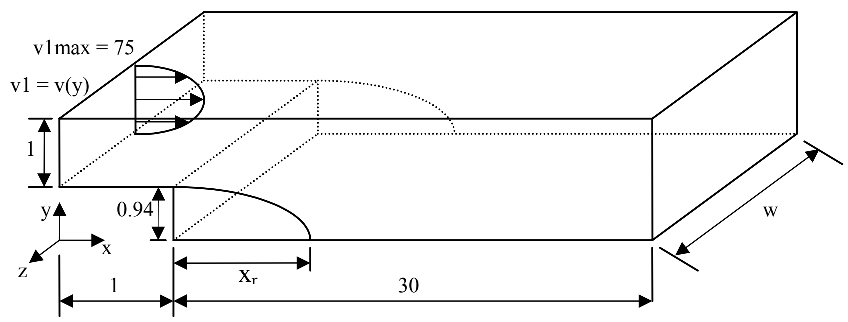

Figure 2 – Domain and characteristics dimensions of the backward-facing step.

For problems characterized by Reynolds number 1000 or less, the time periods of the simulations are long enough to get the stationary average flow. To eliminate spurious pressure oscillations, it was necessary to take very small time intervals, leading to very high processing time.

As inflow boundary conditions, a completely developed parabolic velocity profile was used (v1=V(y), v2=0) at the entrance and no-slip condition (v1=v2=v3=0) were prescribed at the upper and lower walls. At the outflow natural boundary conditions exists (t_1=t_2=t_3=0) (see equation 5). Homogeneous initial conditions (v1=v2=v3=p=0) were used, in the first simulation, for Re=100. In the other simulations the last fields of pressures and velocities calculated for Reynolds number 100 were used.

The Reynolds number is defined in the same way that in the experiments of Armaly et al, 1995..

$$\mathrm{Re} = \frac{\rho\left(2V\max/3\right)2h}{\mu} \tag{23}$$

To define the reattachment length it was investigated the first layer of nodes above the lower wall behind the step, the reattachment point is defined as the first node of the mesh, after the separation region (figure 2), where the component v_1 of the average velocity field assumes a positive value. The adimensional reattachment length is defined as Xr/H, where H is height of the step (0.94m). The results for the relation Xr/H x Re obtained at the present work, together with the values presented in Armaly, et al., 1983, are shown in Table 4.1.

The results for the flows with Re=100 and Re = 400 are in good agreement with experimental data, for the two models. For laminar flows no important difference between the results was expected. The dynamic model for Re=100 presented 95% of the nodes with C(x, t) equal to zero, the flow with Re=400 presented 94% of the nodes with C(x,t) equal to zero and there are not elements having the limited negative eddy viscosity. For the flow with Re=1000, 60% of the nodes presents dynamic coefficient equal to zero, while 0.09% of the elements had limited negative eddy viscosity.

Table 4.1 - Reattachment length ,Xr/H, as a function of the Reynolds number, present work x experimental work.

.

X_r/H x Re			
Re	**Smagorinsky´s Model**	**Dynamic Model**	**Armaly et al. 1983 (exp.)**
100	2.74	2.89	3.0
400	7.60	7.90	8.0
1000	11.09	11.25	16.0

For Re=1000, the numerical result is far from the experimental work, this is an expected error because in two-dimensional simulations of laminar flows, characterized by Reynolds numbers up to 500 the reattachment length is sub-estimated. The error is due to the importance of three-dimensional effects (Silveira Neto et, 1993, and Kaiktsis et al, 1991) and is verified by the numerical experiments of Williams and Baker, 1997.

For the simulation of a turbulent flow, the Reynolds number 10,000 was used.

At the instant of time 0,4 s, using the dynamic model, the re-attachment lengh was of 7.3, when the published experimental values are 7±1 (Kim et al, 1980). The number of elements where the limitation of negative eddy viscosity was applied was of less than 6%.

Figure 3 presents the instantaneous distribution of the vorticity in the field of flow simulated with the dynamic model and Reynolds number equal to 10,000.

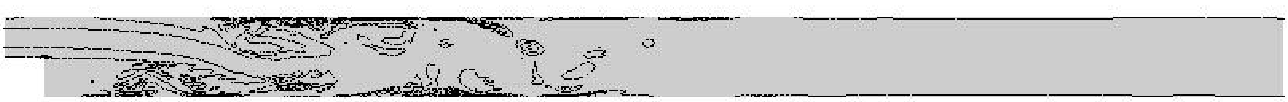

Figure 3 – Vorticity field. Results obtained with te dynamic models, for Re = 10,000.

Finally, results for simulations obtained with the dynamic model, for the three-dimensional flow over a backward facing step, without sidewalls, for Reynolds number 100 and 1,000, are presented. The problem domain is presented in Figure 2, where w=2m. The adopted mesh is similar to that employed for the previous two-dimensional simulations, but with 8 elements in the x_3 direction. Boundary conditions are similar to those used in the two-dimensional case. In the first simulation it was used homogeneous initial conditions and in the following simulation the last fields of pressures and velocities calculated for the Reynolds number 100 are used as initial conditions.

As for the two-dimensional example, flow with Re=100, the obtained reattachment length was equal to 2.88. For the flow with Re=1,000, instantaneous results can be observed in Figures 4 and 5. Despite to the low resolution in the x_3 direction, where only 8 elements are taken, the partial results are qualitatively coherent with the results found in references (Silveira Neto et al., 1993; Lesieur, 1999 and Williams and Baker, 1997). The figure 4 and 5 show that the flow is developing as expected.

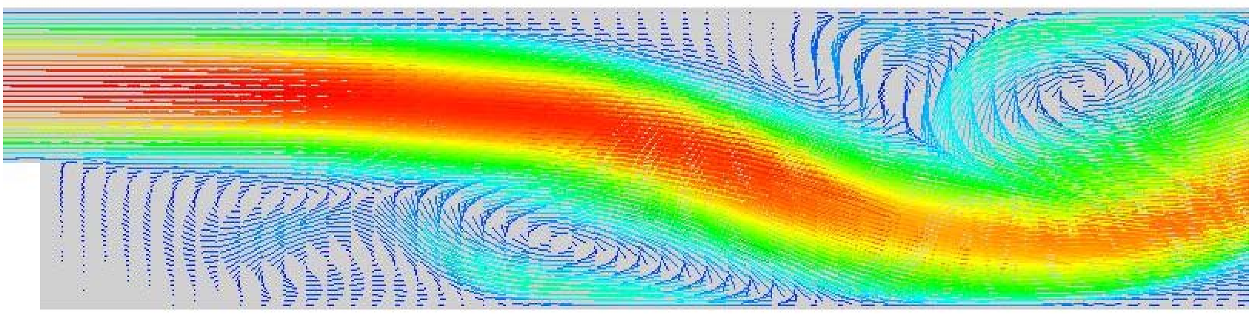

Figure 4- Detail of the velocity vectors, at the center longitudinal section, for the three-dimensional case with the dynamic model and Re=1,000,

4. Conclusions

A methodology to solve quasi-incompressible, turbulent, three-dimensional, isothermal and transient flow of Newtonian fluids, using the Finite Element Method and Large Eddy Simulation, with the Smagorinsky's and the eddy viscosity dynamic model, was presented. A new scheme for the second filtering operation in the dynamic model was developed and applied. The dynamic model has an additional cost in processing time between 9 and 18%, when compared to the implemented Smagorinsky´s model. This additional time is in the same order of the best resultas reported by other authors.

Results are coherent with the experimental works and other numerical. However, the analysis of the problems presented here demonstrated the limitation of the code application to large computational problems with high Reynolds numbers. It was verified that a very small integration time step must be used to prevent spurious oscillations of pressure. The severe restriction on the time step led to extreme expensive time processing, which indicates the necessity to improve the code before continuing the studies on the simulation of more complex problems, as well as comparative studies between the different models.

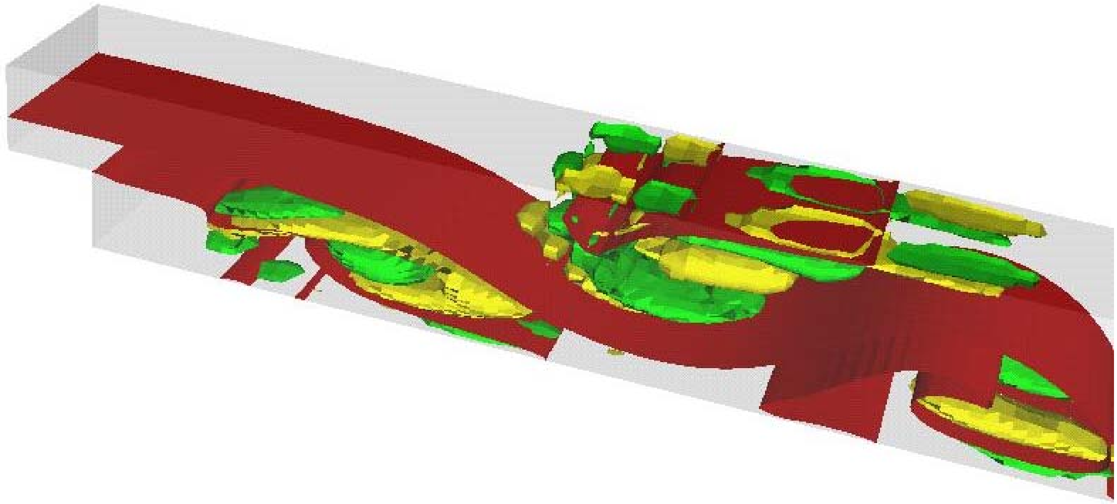

Figura 5 –Vorticity (**w**), with Re=1,000, using the dynamic model: ♦- w_3=5.7x10^1, ♦-w_1=4x10^{-4} e ♦ - w_1= -4x10^{-4}.

To improve the code performance on high Reynolds problems it is necessary to overcome the time step reduction, which is necessary to avoid pressure spurious oscillations. Researches on this are in progress. To analyze the performance of different subgrid scale models, more complex and larger computational problems with refined meshes must be simulated. Finally, there are studies in progress about the quality of three-dimensional computations for LES applying an higher order Taylor-Galerkin scheme (Colin and Rudgyard, 2000).

5. Acknowledgements

The authors thank the National Supercomputer Center on the Federal University of Rio Grande do Sul.

6. Bibliographic References

Armaly, B.F., Durst, F., Pereira, J.C.F., Schönung, B., 1983**.** “Experimental and Theoretical Investigation of Backward-Facing Step Flow”, **Journal of Fluid Mechanics**, v. 127, pp. 473-496.

Awruch, A.M., Petry, A.P., 1997. “**On the Analysis of Viscous Incompressible Flows Using Finite Elements with the Primary Variables” (in Portuguese),** Research Report - RP 122/97, . Programa de Pós-Graduação em Engenharia Civil, Universidade Federal do Rio Grande do Sul, Porto Alegre, RS, Brazil, 73 pages.

Azevedo, R. L. 1999. “**Analysis of Fluid-Structure Interaction Problems Using the Finite Element Methot and Monolithic Coupling**”(in Portuguese). DSc Thesis . Programa de Pós-Graduacão em Engenharia Civil, Universidade Federal do Rio Grande do Sul, Porto Alegre, RS Brazil.

Brasil Junior, A.C.P., 2000, “Finite Element Method Applied to Turbulence”(in Portuguese), **Turbulence: Proceeding of the First Spring School in Transition and Turbulence**, Editors: Freire, A.P.S. ,. Menut P.P.P e Su, J., ABCM, Rio de Janeiro, Brazil.

Burbridge, H.P., Awruch, A.M., 2000. **An efficient one-step Taylor-Galerkin Scheme to Analyze 3D High Compressible Flows Using the Finite Element Method**, Anais ENCIT 2000, Porto Alegre, RS,Brasil

Colin , O., Rudgyard, M., 2000. “Development of High-Order Taylor-Galerkin Schemes for LES”, **Journal of Computational Phisics**, 162, 338-371.

Donea, J, 1984. “A Taylor-Galerkin Method for Convective Transport Problems”, **International Journal for Numerical Methods in Engineering,** v 20, pp 101-119.

Ferziger, J.H.,1993. “Simulation of complex turbulent flow: recent advances and prospects in wind engineering”, **Journall of Wind Engineering and Industrial Aerodynamics**, v.46 & 47, p.195-212, Elsiever.

Findikakis, A.N.; Street, R.L., 1982. “Mathematical Description of Turbulent Flows”, **Journal of Hydraulics Division**, ASCE, V108, No.HY8, paper 17265, p887-903.

Germano, M. Piomelli, U., Moin, P. CabotW.H., 1991. “A dynamic sub-grid-scale eddy viscosity model”, **Phys.Fluids** A3 (7), 1760-1765.

Grötzbach, G. , 1987. "**Direct numerical and large eddy simulation of turbulent channel flows"**, Encyclopedia of Fluid Mechanics, vol 6, Gulf, Houston, pp. 1337-1391.

Hinze, J.O., 1975. **Turbulence**, McGraw-Hill, New York, 2nd ed.

Hughes, T. J. R., 1987. **The Finite Element Method**, Prentice-Hall, New Jersey.

Kaiktsis, L., Karniadakis, G., Orzag, S.A., 1991. “Onset of three-dimensionality, equilibria, and early transition in flow over a Backward-Facing Step”, **Journal of Fluid Mechanics**, v. 231, pp. 501-528.

Kawahara, M., Hirano, H.,1983. “A Finite Element Method for High Reynolds Number Viscous Fluid Flow Using Two Step Explicit Scheme”. **International Journal for Numerical Methods in Fluids**, V.3, pp.137-163.

Kim, W., Menon, S., 1999. “An Unsteady Incompressible Navier-Stokes Solver for Large Eddy Simulation of Turbulent Flows”. **Int. J. Numer. Meth. Fluids**, v. 31, pp. 983-1017.

Lesieur, M. Comte, P., Métais, O.,1995. “Numerical simulations of coherent vortices in turbulence”, **ASME - Appl Mech Rev**, v 48, n 4.

Lilly, D.K., 1992. “A proposed moddification of the Germano subgrid-scale closure method”, **Phys.Fluids A4** (3), 633-635.

Oshima, M., Kobayashi, T., Taniguchi, N., Tsubokura, M., 1996. “Development of Filtering Operation for Dynamic SGS Model Using Finite Element Method**”. The Second ERCOFTAC Workshop on Direct and Large Eddy Simulation.** Septembre 16-19, 1996. Grenoble, Franca.

Padilla, E. L. M., Silveira Neto, A., 2000. “Numerical Implementation of the Dynamic Model for Turbulent Flows in Cylindrical Coordinates” (in Portuguese). **IV Simpósio Mineiro de Mecânica Computacional.** Uberlândia, MG, Brazil. pp. 573-580.

Petry, A.P., 2002. “**Numerical Analysis of Tree-Dimensional Turbulent Flows Using the Finite Element Method and Large Eddy Simulation**”(in Portuguese), DSc. Thesis. Programa de Pós-Graduacão em Engenharia Mecânica, Universidade Federal do Rio Grande do Sul, Porto Alegre, RS Brazil.

Petry, A.P., Awruch, A.M., 1997. “Turbulent Flows Simulation Using the Finite Element Method and Large Eddy Simulation”(in Portuguese), **ENIEF 97,** Proceedings, San Carlos de Bariloche, Argentina.

Reddy, J.N., Gartling, D.K.. 1994. **The Finite Element Method in Heat Transfer and Fluid Dynamics**, CRC Press.

Reuther,J., Alonso,J.J., Rimlinger,M.J., Jamenson , A., 1999, “Aerodynamic shape optimization of supersonic aircraft configurations via adjoint formulation on distributed memmory parallel computers”, **Computers and Fluids**, 28: 675-700.

Rogallo, R.S.; Moin, P., 1984. **Numerical Simulation of Turbulent Flows**, Ann. Rev. Fluid Mechanics, v 16 p.99-137.

Schlichting, H. , 1968. "**Boundary Layer Theory**", McGraw-Hill, New York, 1968.

Silveira Neto, A. 2000, “Fluid Turbulence Fundamentals**”, Turbulence: Proceeding of the First Spring School in Transition and Turbulence**, Editors: Freire, A.P.S. ,. Menut P.P.P e Su, J., ABCM, Rio de Janeiro, Brazil.

Silveira Neto, A., Grand, D., Métais, O., Lesieur,M., 1993. “A Numerical Investigation of Coherent Vortices in Turbulence Behind a Backward-Facing Step”, **Journal of Fluid Mechanics**, v. 256, pp. 1-25.

Smagorinsky, J. 1963. “General Circulation Experiment with the Primitive Equations. I the Basic Experiment”, **Monthly Weathther Review**, v. 91,n. 3, pp 99-164.

Tenekes, H.; Lumley,J.L., 1972. **A First Course in Turbulence**, MIT Press.

White, F.M., 1974. **“Viscous Fluid Flow”**, McGraw-Hill, New York.

Williams,P.T., Baker, A.J., 1997. “Numerical Simulations of Laminar Flow over a 3D Backward-Facing Step”, **International Journal for Numerical Methods in Fluids**, v.24, pp. 1159-1183.

Zang, Y., Street, R. L., Koseff, J. R., 1993. “A Dynamic Mixed Subgrid-Scale Model and its Application to Turbulent Recirculating Flows”. **Phys. Fluids A**, v. 5, n. 12, pp. 3186-3196.