NUMERICAL SIMULATION OF A 2D TURBULENT FLOW CLOSE TO A SEPARATION POINT

Leon Matos R. de Lima, matosleon@gmail.com
Norberto Mangiavacchi, norberto.mangiavacchi@gmail.com
UERJ - Rio de Janeiro State University
Rua S F Xavier 524 - Maracanã, Rio de Janeiro - RJ, CEP20550-013, Brazil

Abstract. *This work presents a 2D analysis of a turbulent flow over a steep hill, without turbulence model, focusing on the turbulent boundary layer separation. The numerical methodology employs a Finite Element code with first order Semi-Lagrangian scheme for time discretization. Velocity and pressure fields are decoupled by the Discrete Projection Method. The separation and reattachment points, mean velocity profiles and the wall shear stress distribution are compared to numerical and experimental results found in recent literature. The study provides interesting results regarding the applicability of simple 2D numerical models to the simulation of turbulent flows over hills.*

Keywords: *Turbulent flows, Finite Element Method, boundary layer separation.*

1. INTRODUCTION

Most relevant flows in nature or in man made constructions are turbulent. However, despite its relevance, there is still much to know about turbulent flows. Indeed, as pointed by some authors, these flows are the last open problem remaining from classical mechanics (Moriconi, 2008). In particular, the boundary layer separation in such flows is the subject of many recent works. For example, the work of Simpson (1995) presents some fundamental concepts regarding turbulent boundary layer separation, and Loureiro (2008) provides an extensive review of the state of the art of this subject.

In particular, turbulent flows over hills are a relevant problem in fluid dynamics because it directly represents many environmental flows, mostly in the micrometeorology field, and also because properties of the flow after the hill are important for a number of fluid dynamics applications. For these reasons, many recent works had been devoted to the prediction of flow over surfaces that generates boundary layer separation. Most of them presents numerical simulations. Nevertheless, the appropriate approach for adequately addressing the turbulence properties in such flows is not yet well established. Regarding this, the work of Loureiro *et al.* (2008) presents a detailed study on the applicability of six different turbulence models – four eddy-viscosity models and two Reynolds stress models. In that work, results were validated with the experimental data of Loureiro *et al.* (2007).

This paper presents a study of a turbulent flow over a steep hill through a two-dimensional approach without turbulence models. Results will be compared to those found by Loureiro *et al.* (2007) and Loureiro *et al.* (2008). The objective is to evaluate the capability of a simple 2D numerical model to correctly predict the turbulent flow. The methodology employed consists of a Finite Element Method code, with Galerkin method for space discretization and a first order semi-Lagrangian scheme for the discretization of the convection term (Wiin-Nielsen, 1959).

In what follows, section 2 makes a brief description of the numerical model employed in this work, section 3 presents some aspects concerning the finite element mesh, section 4 brings the results obtained from the simulations carried in this work, and section 5 presents the conclusions of this paper.

2. NUMERICAL MODEL

The mathematical model consists of the incompressible Navier-Stokes equations with constant viscosity, expressed by

$$\nabla \cdot \boldsymbol{v} = 0 \tag{1}$$

$$\frac{\mathrm{D}\boldsymbol{v}}{\mathrm{D}t} = -\frac{1}{\rho}\nabla p + \nu \nabla^2 \boldsymbol{v} \tag{2}$$

where $\boldsymbol{v}$, p, ρ and ν are the velocity and pressure fields, the specific mass and the kinematic viscosity, respectively. Equations (1) and (2), together with the proper boundary conditions, are numerically solved by the Finite Element Method, where the velocity and pressure fields are decoupled by the discrete Projection Method. As previously mentioned, space domain is discretized by the standard Galerkin method, and time domain is discretized by a first order semi-Lagrangian method, which combines aspects of classical eulerian and lagrangian schemes, and has the advantage of eliminating the restriction on time step concerning numerical stability. On the other hand, depending on mesh resolution, semi-Lagrangian schemes may lead to inaccurate results in flows with streamlines of accentuated curvature (Silva, 2011). Another aspect of the semi-Lagrangian method is that it increases numerical diffusion. For this study in particular, this characteristic carried on the effects of momentum dissipation in the transverse direction, not considered in the 2D model. Mini elements are employed for velocity field, and linear elements for pressure field. This procedure leads to two algebraic linear systems, one for velocity (which is solved by the Conjugate Gradient Method) and another for pressure (solved by the Generalized Minimum Residue Method (Saad and Schultz, 1986)). Further details on the numerical formulation can be found in Shin (2009), Lima (2010) and Lopez (2010).

3. FINITE ELEMENT MESH

Spatial domain is discretized by a triangular finite element mesh. The generation process starts from a structured grid, where the vertical distance between the points decreases exponentially as they get closer to the wall, which provides better refinement at the boundary layer region.

In order to reproduce the geometry employed by Loureiro *et al.* (2008), the curved surface at the bottom follows a modified "Witch of Agnesi" profile. In the present text, x denoted the longitudinal direction, and z is the vertical direction. Let x_b be the set of x values where hill is defined, i.e., $x_b = \{x \in \mathbb{R} | 2L_h \leq x \leq 2L_h\}$, where L_h is the characteristic length of the hill, let H_1 and H_2 be such that $H = H_1 - H_2$, where H is the maximum height of the hill, and let z_b be the hill elevation. The profile is then given by

$$y_b = \frac{H_1}{1 + (x_b/L_h)^2} - H_2 \tag{3}$$

The domain is $4H$ high and $24H$ long. For this problem, $H = 6$mm, $H_2 = 1.5$mm and $L_h = 2.5H$. Figure 1 shows the en

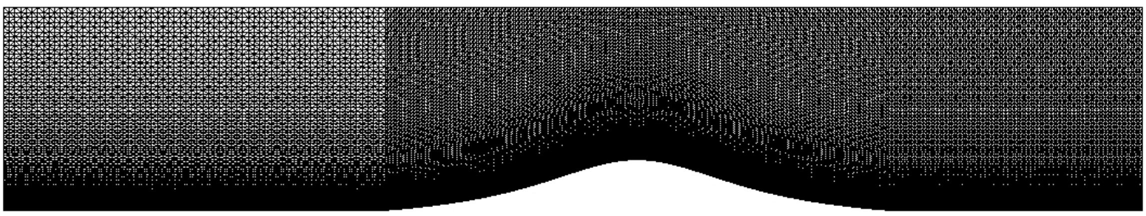


Figure 1. Finite Element Mesh on the simulated domain.

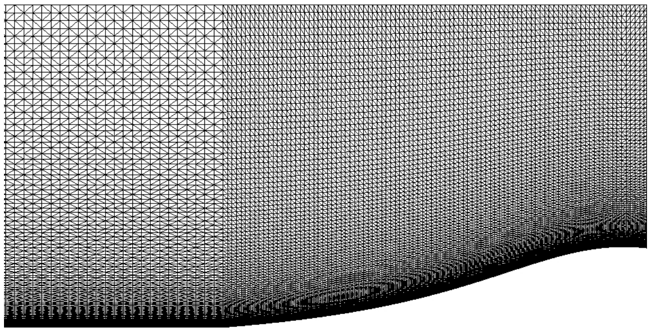


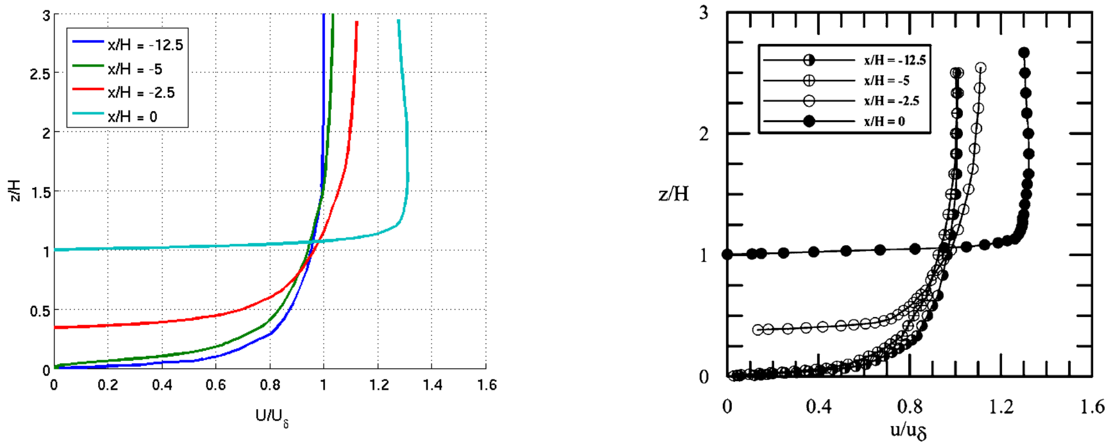
Figure 2. Zoom of the Finite Element Mesh on the hill region.

It can be noted that, besides the vertical refinement, there is a longitudinal refinement after the beginning of the hill.

4. RESULTS AND DISCUSSION

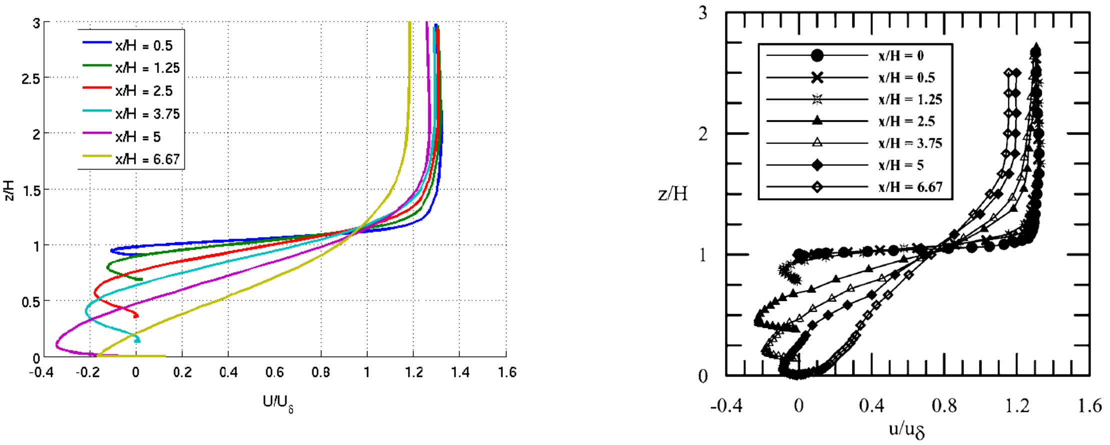
The following results were obtained from simulations performed on a mesh of 33,734 nodes and 66,600 elements. Calculations were interrupted at time step 10,000, and the velocity field were averaged from time steps 3,000 to 10,000. The simulation predicted boundary layer separation at $x/H = 0.153$ and reattachment at $x/H = 7.348$ (for u taken at $z/H = 0.011$). In the experiment of the reference paper Loureiro *et al.* (2008), reattachment occurred at $x/H = 6.67$.

Mean velocity profiles from 13 different sections will be presented in the next figures, together with the experimental results of Loureiro *et al.* (2007).



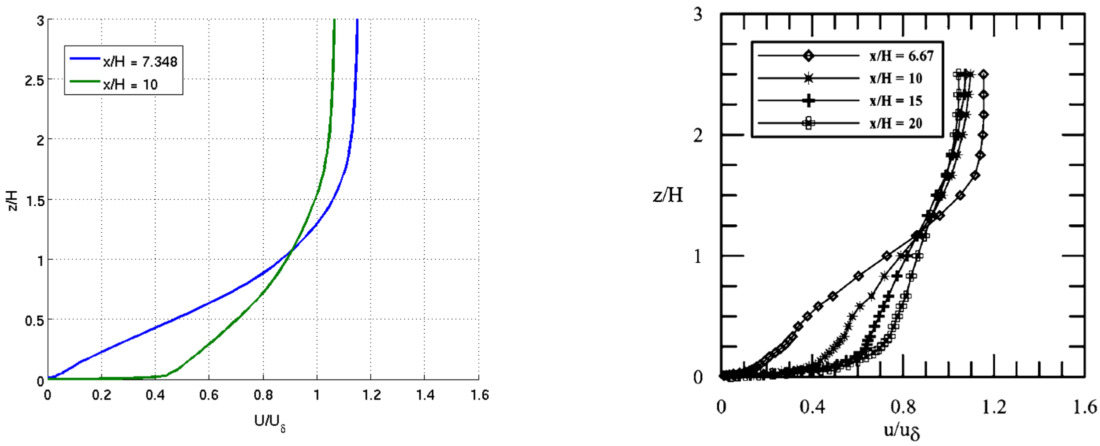
(a) (b)

Figure 3. Mean velocity profiles for x/H =-12.5, -5, -2.5 and 0.



(a) (b)

Figure 4. Mean velocity profiles for x/H =0.5, 1.25, 2.5, 3.75, 5 and 6.67.



(a) (b)

Figure 5. Mean velocity profiles for x/H =7.348 and 10.

At points upstream the separation point, the simulated velocity profiles show good agreement with respect to the experiment. After separation, results are still close to those of the experiment until section $x/H = 5$. There is a considerable difference between the profiles at section $x/H = 6.67$.

The following figures present comparisons between the simulated results and the simulations performed by Loureiro *et al.* (2008). In that work, four eddy viscosity models – $\kappa - \epsilon$, $\kappa - \omega$, SST and RNG – and two Reynolds stress models – SSG and BSL – were employed. Details on each of these models can be found in the same work. Velocity profiles are compared at sections $x/H = 0$, $x/H = 3.75$ and $x/H = 10$.

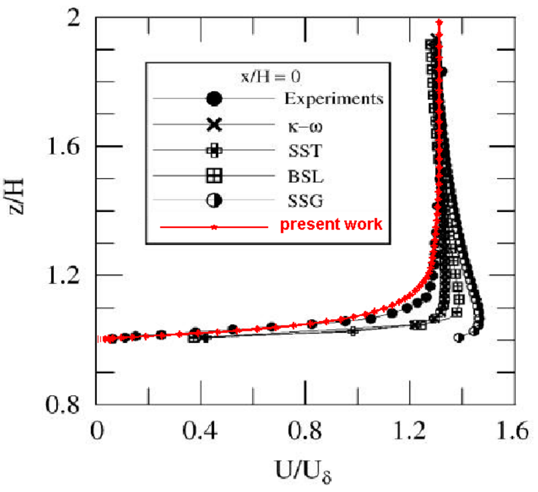


Figure 6. Mean velocity profiles at $x/H = 0$.

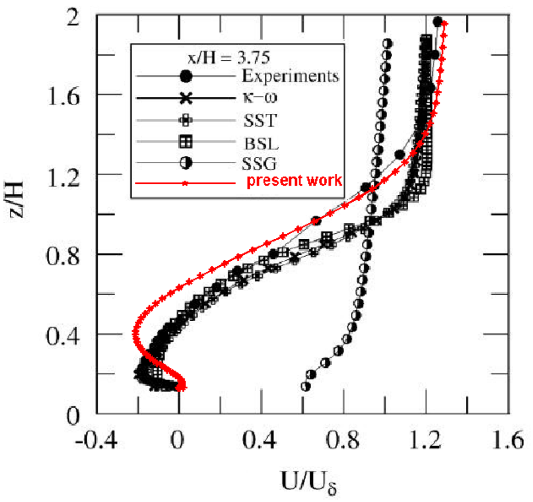


Figure 7. Mean velocity profiles at $x/H = 3.75$.

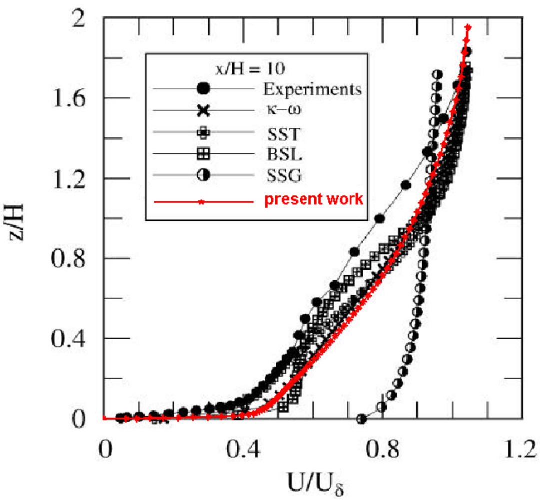


Figure 8. Mean velocity profiles at $x/H = 10$.

As pointed by Loureiro *et al.* (2008), the SSG provided very inaccurate results compared to the other methods. The profiles obtained in the this work present a slightly different behavior with respect to those obtained from the turbulence models, and show better agreement with the experiment in the region upstream the separate flow.

Another property of the flow, the wall shear stress (τ_*), have also been computed (fig. 9). Upstream the separation, the 2D model was the only that did not overpredicted the wall shear stress. After $x/H = 0$ the 2D model provided worse results compared to the $\kappa - \omega$, the SST and the BSL models, although better than the $\kappa - \epsilon$, the RNG and the SSG models.

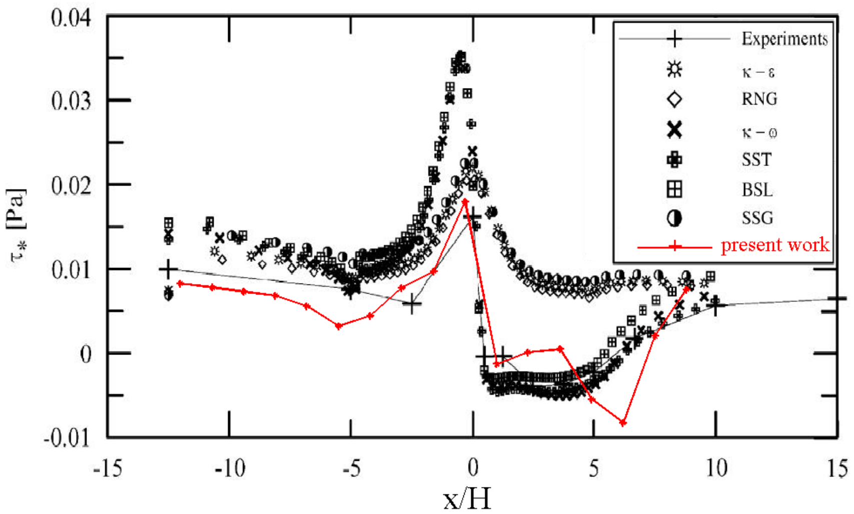


Figure 9. Wall shear stress distribution.

5. CONCLUSIONS

The objetive of this paper was to test the performance of a direct 2D finite element code in the simulation of a relevant turbulent flow, without the use of turbulence models. Somewhat surprisingly, the results showed good agreement to the reference experimental data, and more accurate than some of the turbulence models tested by Loureiro *et al.* (2008). Though further investigation is required, one of the possible reasons for this is the dissipative character of the semi-Lagrangian scheme. It can be noted that all the velocity profiles obtained from the codes with turbulence models were overpredicted with respect to the experiment in most of the sections, i.e., the velocity values were essentially larger in the simulations with turbulence models than in the experiment. The results suggest that, for such a 2D turbulent flow, with low Reynolds number, a direct 2D simulation may provide accurate predictions.

6. REFERENCES

Lima, L.M.R.D., 2010. *Desenvolvimento de Modelos 2D Para Simulação de Escoamentos Ambientais*. Master's thesis, PPG-EM/UERJ.

Lopez, P.J.T., 2010. *Parallel Implementation of Finite Element Code for Two-Dimensional Incompressible Navier-Stokes Equations With Scalar Transport*. Master's thesis, PPG-EM/UERJ.

Loureiro, J.B.R., 2008. *Escoamento Turbulento sobre Colinas Abruptas Lisas e Rugosas com Extensas Regiões de Separação*. Tese de doutorado, COPPE/UFRJ, Rio de Janeiro - RJ.

Loureiro, J.B.R., Alho, A.T.P. and Freire, A.P.S., 2008. "The numerical computaion of a near-wall turbulent flow over a steep hill". *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 96, pp. 540–561.

Loureiro, J.B.R., Pinho, F.T. and Freire, A.P.S., 2007. "Near wall characterization of the flow over a two-dimensional steep smooth hill". *Experiments in Fluids*, Vol. 42, pp. 441–457.

Moriconi, L., 2008. "Desafios da turbulência". *Ciência Hoje*, Vol. 43, pp. 38–44.

Saad, Y. and Schultz, M.H., 1986. "Gmres: A generalized minimum residual algorithm for solving nonsysmmetric linear systems". *Journal of Scientific and Statitical Computing*, Vol. 7, pp. 856–869.

Shin, H.H., 2009. *A Methodology of Study of Three Dimensional Stratified Turbulent Fluid Flow For Hydroelectric Power Plant Reservoir Simulation*. Master's thesis, PPG-EM/UERJ.

Silva, H.C., 2011. *Finite elements in convection dominated flows: a semi-Lagrangian method*. Master's thesis, PPG-EM/UERJ.

Simpson, R.L., 1995. "Aspects of turbulent boundary-layer separation". *Prog. Aerospace Sciences*, Vol. 32, pp. 376–421.

Wiin-Nielsen, A., 1959. "On the application of trajectory methods in numerical forecasting". *International Meteorological Institute in Stockholm*, Vol. 11, pp. 180–196.

7. RESPONSIBILITY NOTICE

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