DIRECT 2D FINITE ELEMENT SIMULATION OF TURBULENT FLOW OVER A STEEP HILL

Leon Lima[1] & Norberto Mangiavacchi [1]
[1]*Mechanical Department - Rio de Janeiro State University (UERJ), 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.

NUMERICAL MODEL

Since the methodology employed in this work does not include any turbulence model, the equations set consists only of the incompressible Navier-Stokes equations with constant viscosity, which 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. 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. Further details on the numerical formulation can be found in [5, 1, 2]. Results will be compared to those found by [4, 3]. The objective is to evaluate the capability of a simple 2D numerical model to correctly predict the turbulent flow. Spatial domain is discretized by a triangular finite elements mesh (fig.).

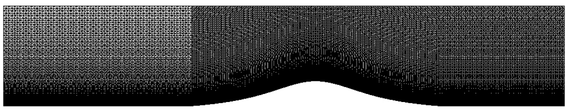

Figure 1. Domain of the problem discretized by a finite elements mesh.

In order to reproduce the geometry employed by [3], the curved surface at the bottom follows a modified "Witch of Agnesi" profile [3]. In the present text, x denotes the longitudinal direction, and z is the vertical direction. Let x_b be the set of x values where the 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 $y_b = \frac{H_1}{1+(x_b/L_h)^2} - H_2$. The domain is $4H$ high and $24H$ long. For this problem, $H = 6$mm, $H_2 = 1.5$mm and $L_h = 2.5H$.

RESULTS AND DISCUSSION

The following results were obtained from simulations performed on a mesh of 33,734 nodes and 66,600 elements. 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 [3], reattachment occurred at $x/H = 6.67$.

At points upstream the separation point, the simulated velocity profiles show good agreement with 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 [3]. 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 $x/H = 10$.

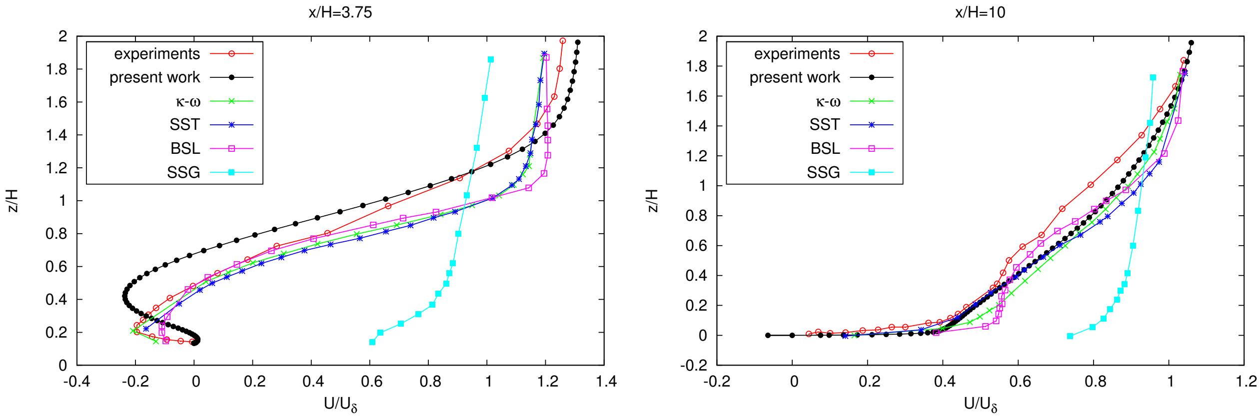

Figure 2. Mean velocity profiles at $x/H = 3.75$ (left) and $x/H = 10$ (right).

As pointed by [3], 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. 3). 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.

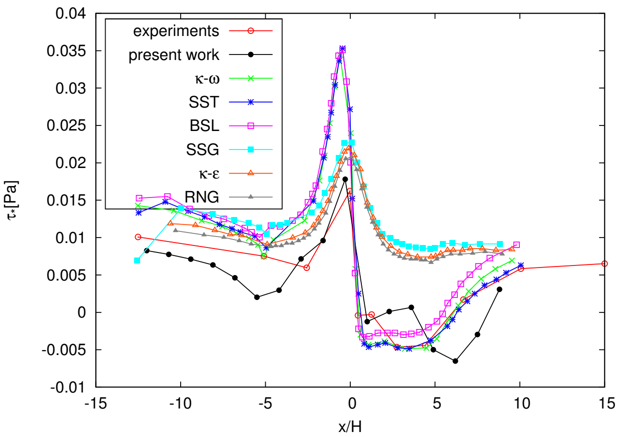

Figure 3. Wall shear stress distribution.

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. Results showed good agreement with the reference experimental data, and more accurate than some of the turbulence models tested by [3] concerning the mean fields. Though further investigation is required, one of the possible reasons for this is the dissipative character of the semi-Lagrangian scheme. The results suggest that, for such a 2D turbulent flow, with low Reynolds number, a direct 2D simulation may provide accurate predictions.

References

[1] Leon Matos Ribeiro De Lima. Desenvolvimento de modelos 2d para simulação de escoamentos ambientais. Master's thesis, PPG-EM/UERJ, Março 2010.

[2] Pedro Juan Torres Lopez. Parallel implementation of finite element code for two-dimensional incompressible navier-stokes equations with scalar transport. Master's thesis, PPG-EM/UERJ, Março 2010.

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

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

[5] Hyun Ho Shin. A methodology of study of three dimensional stratified turbulent fluid flow for hydroelectric power plant reservoir simulation. Master's thesis, PPG-EM/UERJ, Março 2009.