NUMERICAL SIMULATION OF THE FLOW OVER A BACKWARD-FACING STEP IN A BEOWULF-CLASS CLUSTER

Rubens Campregher*, Aristeu da Silveira Neto*, Wellington Marinho*, Sérgio S. Mansur**
*Federal Univ. of Uberlandia, School of Mech. Engineering, 38400-902 Uberlandia - MG, Brazil
**Sao Paulo State University, Dept. of Mech. Engineering, 15385-000 Ilha Solteira - SP, Brazil

Summary This work proposal is to present a three-dimensional numerical simulation of the classical flow over a backward-facing step in a very fine mesh. The in-house parallel numerical code was written in Fortran 90 and has solved the Navier-Stokes Equations discretised by the FVM and has run in a Beowulf-class cluster of PCs. The flow Reynolds number was 38000, a LES turbulence modelling technique was employed with Smagorinsky's subgrid model. The results were compared against the literature.

INTRODUCTION

The parallel computing techniques have gaining importance since the 90's as the modern parallelization libraries has evolved and, as long as can be foreseen, they'll be in a continuing growth [3]. It is worth noting that the processing capabilities increasing rates of mainframes and supercomputers have been far from beat the PC's rates. In addition, the new network hardware and software resources have pushed up the parallelization approaches that employ ordinary PCs. Among the several parallel techniques using clusters of PCs, the probably most renowned one is the Beowulf–class cluster, developed at NASA in 1994 [3] and employed in this present work. Some of the main features, presented in a cluster of computers that follow the Beowulf "philosophy" must have are: the independence of software and hardware suppliers, scalable peripherals, and free open source codes with minimum changes. All of these requirements meet the needs of most universities, more remarkably in those from developing countries, which very often face funding restrictions.
The cluster is composed by five 2.8GHz clock and 1GByte DDR RAM machines connected by a Gigabit network. The message-passing system employed was the MPI, more specifically the MPICH implementation. The numerical code has been written in Fortran 90.
In the field of the Computational Fluids Dynamics (CFD), powerful computer capabilities are frequently requested, because as the flow regime increases, its structures become smaller, requiring also smaller grid sizes in order to reproduce them properly. Consequently, greater computational resources are needed to store and perform the operations on the flow parameters. In this context, the parallel computing in cluster of PCs is suitable.
The flow over a backward-facing step, despite of its simple geometry, produces very rich structures involving a great number of physical mechanisms representative of the separated internal flows [2]. So, this kind of configuration is probably the best numerically documented, and experimentally as well [1]. This work aims to present the successful employment of an in-house cluster of ordinary PCs running an in-house parallel numerical code for evaluating the flow over a backward-facing step, comparing it against the data in the literature [1,2].

NUMERICAL PROBLEM

The flow dynamics can be very well determined, in most of the engineering applications, by the solution of the system generated by the momentum and continuity equations, known as Navier-Stokes Equations:

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

$$\frac{\partial(\rho u_i)}{\partial x_i}=0 \tag{2}$$

The temperature is evaluated by solving the Energy Equation:

$$\frac{\partial(\rho T)}{\partial t}+\frac{\partial(u_j T)}{\partial x_j}=\frac{\partial}{\partial x_j}\left(\alpha\frac{\partial T}{\partial x_j}\right) \tag{3}$$

And the effective viscosity is evaluated by the Smagorinsky's subgrid model [4]:

$$\mu_{eff}=\rho\upsilon_{eff}=\rho(C_s\Delta)^2\sqrt{2S_{ij}S_{ij}} \tag{4}$$

Where C_s is the Smagorinsky's constant, Δ is the mesh size, and S_{ij} is the strain rate of the filtered velocity field [5].

The numerical domain was built as depicted in the Figure 1, and follows the experiments described in [2]. The Reynolds number was due the height H (Re_H), the step span width was three times the W dimension and maximum step length X was 30H.

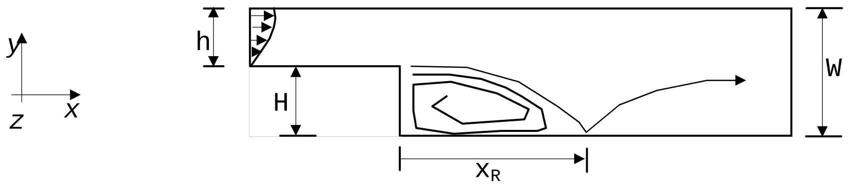

Figure 1 – The numerical domain.

Although the numerical code generates only structured cartesian grid, it is possible to construct a non-uniform mesh in order to concentrate grids in areas where the gradients are stronger, i.e., behind the step and closer to the channel bottom wall. This configuration was done in this work, which allowed saving computer memory. The grid size employed was 740x62x182, i.e., approximately 8.35 millions of cells.
Several flow properties were analyzed, that is, the reattachment length x_R, the velocity profiles (Fig. 2 a,b), the stream and spanwise vorticities (Fig. 3), the Strouhal number (Fig. 2 c) in several points downstream the domain, as well as the energy decay (Fig. 2d).

RESULTS

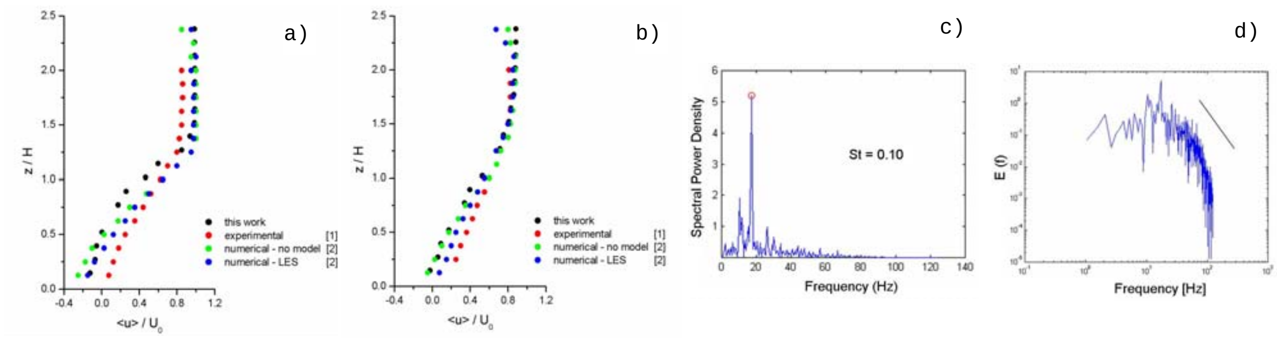

Figure 2 – Velocity profile at x/H = 8 (a), and at x/H = 10 (b), the frequency signal (c), and energy decay (d).

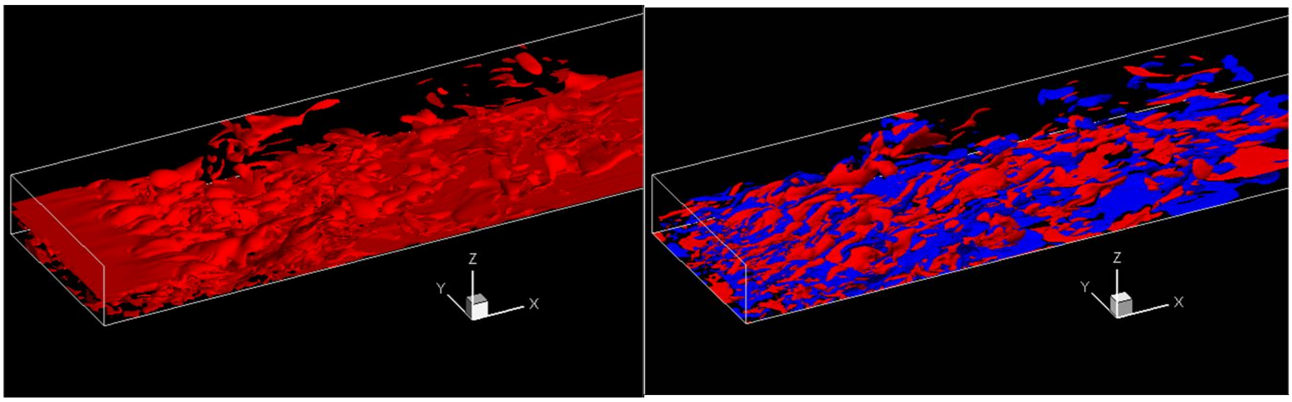

Figure 3 – Isovalues of spanwise vorticity ($\omega_y = 1.8U_\infty/H$) (left) and streamwise vorticity ($\omega_x = 1.5U_\infty/H$).

References

[1] Eaton, J. K. & Johnston, J. P.: "Turbulent flow reattachment: an experimental study of the flow and structure behind a backward-facing step", Stanford University, Rep. MD-39, 1980.

[2] Silveira Neto A., Grand D., Metais O., Lesieur M.: A Numerical Investigation of the Coherent Vortices in Turbulence behind a Backward- Facing Step. *J. Fluid Mechanics*, **256**:1-25, 1993.

[3] Baker, L., Smith, B.J.: Parallel Programming. McGraw-Hill., 1996

[4] Lesieur, M.: Turbulence in Fluids, 3rd Ed., *Kluwer Academic Publishers*, 1997.

[5] Sagaut, P.: Large Eddy Simulation for Incompressible Flows, 2nd Ed., Springer, 2002.