A NUMERICAL TURBULENT WIND TUNNEL FOR LOW REYNOLDS NUMBER FLOWS

P.Orlandi[1], & F.Fabiani
Dipartimento di Meccanica e Aeronautica
Via Eudossiana 16, Roma Italy

The large increase of speed and memory of the computer hardware is at the point that there is the possibility to replace in the near future the wind tunnels with Navier-Stokes numerical simulators. This was an old goal, which was partially done, by simulations in simple geometries, for instance turbulent channels or pipes. For flows past irregular geometries in the past attempts to use boundary fitted coordinates, had success to explain the physics of drag reducing riblets (Orlandi, 1989, Choi *et al.* , 1993). There were, however, difficulties to extend the method to very complex geometries, and to solve the elliptic equations. This was the reason why, the interest moved to flows in orthogonal grids, simple boundaries and complex physics; until a revolution came to treat complex boundaries by the improvements of the immersed boundary method by Fadlun *et al.* (2000). Since then an explosion of this method is in progress, with different versions differing slightly depending on the applications.

For instance the Fadlun's version, which does not solve the Navier-Stokes equations at the points close to the solid boundary, cannot be applied to flows in channels with rough boundaries and constant flow rate. The modification by Leonardi & Orlandi (2004), has been successfully applied to any kind of rough channels with periodic conditions in two directions. This method here has been applied to several kind of flows, including those with inlet outlet boundary conditions which are briefly listed.

a) Flow past grids of different solidity to mimic the laboratory experiments to understand the decay of isotropic turbulence.

b) Flows past bodies close to a solid boundary, to investigate the variations on lift and drag with the distance of the body from the wall. A 3D simulation past a rotating wheel is presented to show the feasibility of the method to study the tire vortex dynamics.

c) Flow past flapping wings, to understand the physics of unsteady lift and drag.

d) Flow past unsteady rough surfaces, to understand whether the unsteadiness reduces or increase the friction. In this case two- and three-dimensional unsteady surfaces are considered.

References

[1] Choi, H., Moin, P. & Kim, J. 1993. Direct Numerical Simulation of Turbulent Flow Over Riblets. *J. Fluid Mech.* **255**, 503–538.

[2] Fadlun, E. A., Verzicco, R., Orlandi, P. & Mohd-Yusof, J. 2000. Combined immersed boundary finite-difference methods for three-dimensional complex flow simulations. *J. Comput. Phys.* **161**, 35–60.

[3] Leonardi S. & Orlandi P. 2004 A numerical method for turbulent flows over complex geometries. *ERCOFTAC bullettin* **62**, 41-46.

[4] Orlandi, P. 1989. Numerical solution of 3-D flows periodic in one direction and with complex geometries in 2-D. *Annual Research Briefs, Center for Turbulence Research 1989*, 215-230.

[1]orlandi@kolmogorov.ing.uniroma1.it