MICROSCALE AIRFLOW MODELLING USING THE IMMERSED BOUNDARY METHOD AND IMPLICIT LES.

Vladimír Fuka, Josef Brechler
Department of Meteorology and Environment Protection, Faculty of Mathematics and Physics, Charles University, Prague

Abstract
This contribution shows some results when the implicit large eddy simulation (ILES) methodology was used and tested in the 2D model of air-flow around an obstacle. The obstacle is a square cylinder of the infinite length. This geometry enables to use the 2D approach that considerably speeds up the computation. Obtained results have been compared with other results of laboratory experiments or 2D and 3D LES numerical simulations.

1. INTRODUCTION
Flow within the atmospheric boundary layer is nearly always turbulent even above a flat and not too rough terrain. The intensity of turbulence increases when terrain is undulated or mountainous or when the flow occurs in some geometrically complicated environment. An example of such kind of environment is, for example, the urban one. Beside of complicated flow pattern with many individual circulation systems a very important role is played by turbulence that appears thanks to enormous shears. These shears are consequences of the interaction of the mentioned circulation systems with one another and with the basic flow and also due to bypassing the obstacles – buildings and other urban structures. If we model airflow in geometrically complicated areas it is important for the turbulence to be described and implemented into the model equations well otherwise we do not obtain a real pattern of flow.

Now there exist two basic approaches how turbulence can be implemented into the Navier-Stokes equations. The first one consists in utilization of the Reynolds averaged Navier-Stokes equations (abbreviated as RANS) and then using some turbulence model (algebraic or k-ε, for example). For the second approach the term large eddy simulation (LES) is used. In this approach resolvable turbulent eddies are solved explicitly by the filtered Navier–Stokes equations and the impact of subgrid turbulence eddies is parametrized by some subgrid model (at this moment some versions of the original Smagorinsky model are used, for example). In the last decade a possible new approach how to model turbulence appeared. This new approach is called implicit LES and abbreviated as ILES. The principles of this method were firstly introduced by Boris, 1989 and his concept came from the monotone numerical algorithms used for LES and called monotone integrated LES and abbreviated as MILES.

The ILES approach together with so called immersed boundary method (Peskin, 1982) that enables to use simple orthogonal grid (Cartesian grid, for example) for complicated geometrical structures and shapes makes the problem easier for application of high resolution methods together with easier coding.

2. METHOD

At this moment the basic set of equations that create the prepared CFD high resolution model for urban areas consists from the following ones: continuity equation and Navier-Stokes equations for non-stratified and incompressible fluid. All the equations are written in the dimensionless form. As a reference length and reference velocity used for normalization of dimensional variables the velocity at the inflow boundary and some characteristics extension of bypassed obstacle have been used. These parameters are used also in the specification of Reynolds or Strouhal numbers. For the spatial discretization the finite volume method has been used. The temporal discretization is made with the fractional step method (Brown et al., 2001).

The method enabling to utilize the ILES approach consists in the application of non-linear scheme for approximation of advective (non-linear) terms. The advective (in hydrodynamic also called convective) terms are computed by means of Kurganov and Tadmor scheme (see Kurganov, E. Tadmor, 2000) with MUSCL type of reconstruction (van Leer, B., 1979). Nonlinearity is put into this scheme via the application of slope limiters assuring that no computational extremes will appear. This consequence of the non-linearity of the used scheme is the other and very important property that was firstly discussed by Boris.1989. His work started the period when there has been an increasing amount of evidence that high resolution numerical methods for hyperbolic partial differential equations have an embedded (or implicit) turbulence model. In this contribution there are shown some preliminary results that deals with the implicit LES utilization. When looking at them and comparing them with the results of other authors or experimental data, it is necessary to have in mind that at this study results of 2D approach are shown and also no wall model in the vicinity of the obstacle has been used,

The results that are presented here correspond to the parameters showed in the Table1, meaning of used symbols is evident from the sketch of the problem geometry. This sketch is shown on Figure 1. The inlet velocity was set to the constant one and also its profile corresponds to the constant one.

Parameter	Value
Reynolds number	5000
Grid resolution near the wall	0.01
Blockage ratio d/H	7.7%
Inlet boundary condition	U = const.

Table 1. Used parameters

The immersed boundary method suggested by Kim et al., 2001, has been applied in this case. Some advantages connected with the utilization of this method consist in:

a) Cartesian grid application,
b) equations written in simple form,
c) effective solver utilization,
d) simple grid generation.

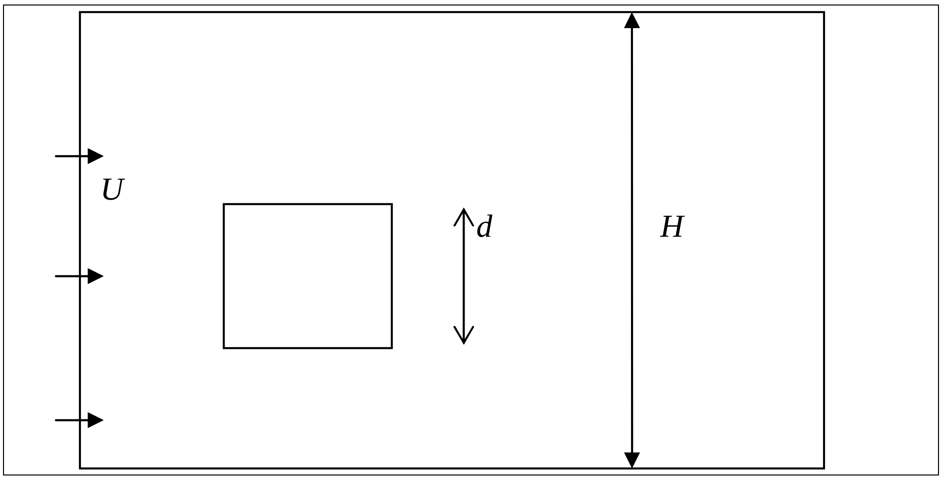

Figure1. Geometry of the problem

3. RESULTS

Results for the above presented geometry and parameters are shown and discussed in this paragraph. When looking at them and discussing them it is necessary to have in mind that 2D approach has been applied and no wall model has been used as it was said above.

On the first figure – Figure 2 - the instantaneous vorticity field can be seen. Dark and white areas correspond to vortices with the reversal sense of rotation.

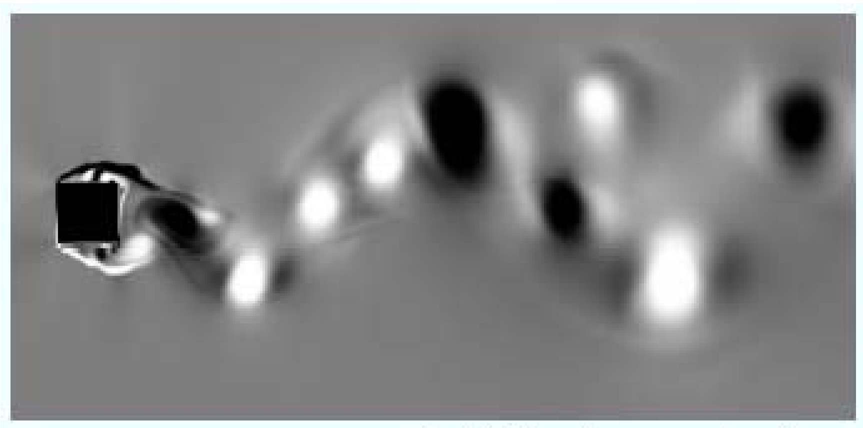

Figure 2. The instantaneous vorticity field behind the square cylinder.

The graph depicted on the Figure 3 shows the time averaged velocity along centreline of square obstacle and the comparison of the model results with other 2D and 3D numerical experiment and with the results of laboratory experiments. On the *X* axis there is the distance measured along the inflow velocity direction. Zero point *x*-coordinate corresponds to the upwind wall of the obstacle, a point with the coordinate *x* is equal to 1 corresponds to the downwind wall of the obstacle. Values shown on the *Y* axis correspond to the dimensionless values of velocity (the actual velocity normalized with the velocity on the inlet boundary). In comparison with the experimental data (black circles or black triangles) we can see from our model results that we got a little bit smaller recirculation zone behind the obstacle and as a consequence of this fact the a little bit higher velocity behind the obstacle. The pattern of the velocity profile corresponds to that of the experimental data. When comparing our model result with those of other 2D or 3D LES simulations we can conclude that with the exception of 3D simulation of Breuer and Porque, 1996 they correspond well to the other simulation result no matter how whether they are 2D or 3D. Only the mentioned result of Breuer and Porque, 1996 fits well to the experimental data of Durao et al., 1988. The improvement of our results could be brought by the wall model utilization as this should improve the flow patter close to the obstacle walls.

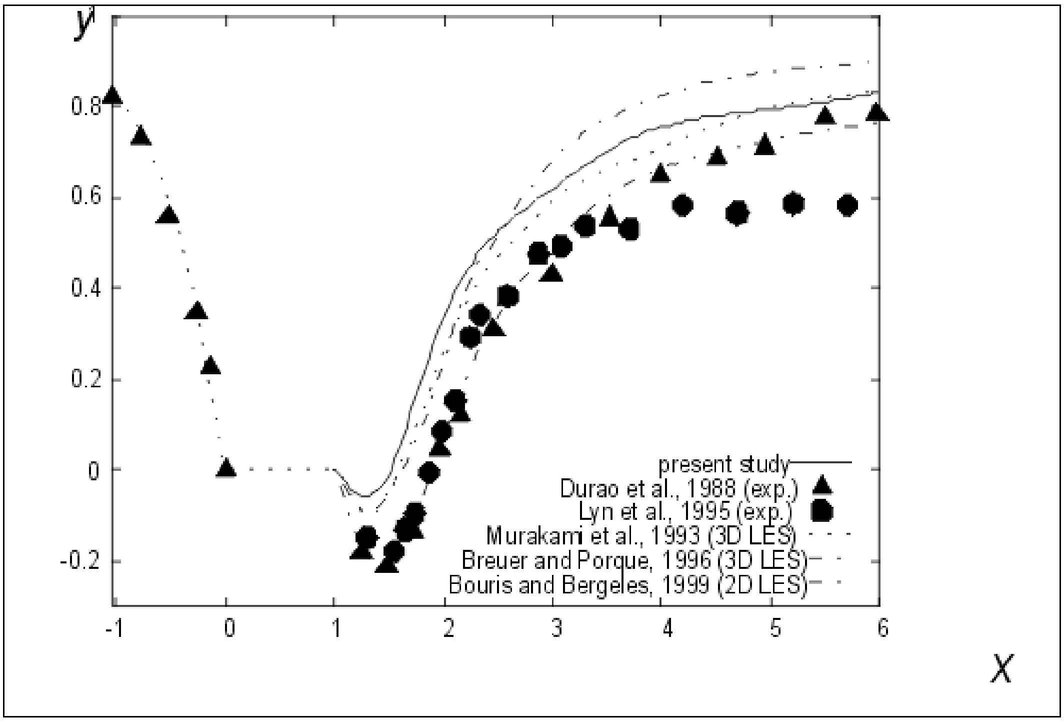

Figure 3. Time averaged velocity along the centreline.

On the last Figure 4 the streamlines corresponding to the time averaged velocity field is shown. The white lines are streamlines, colour or greyscale correspond to the velocity magnitude when the red (or dark) zones on both sides of the upwind wall correspond to the highest velocity areas.

The last information deals with the values of Strouhal numbers. They are presented in the following Table 2.

Reference	**St**
Lyn et al., 1995 - experiment	0.132
Durao et al.,1988 - experiment	0.139
Bouris and Bergeles, 1999	0.134
present study	0.12

Table 2. Comparison of computed St with experiments and other 2D simulation

From the Table 2 it can be seen, that the St is a little bit lower than the one obtained for laboratory experiments or the compared other 2D numerical simulation. We suppose that the main reason for it is the missing wall model that would improve the situation in

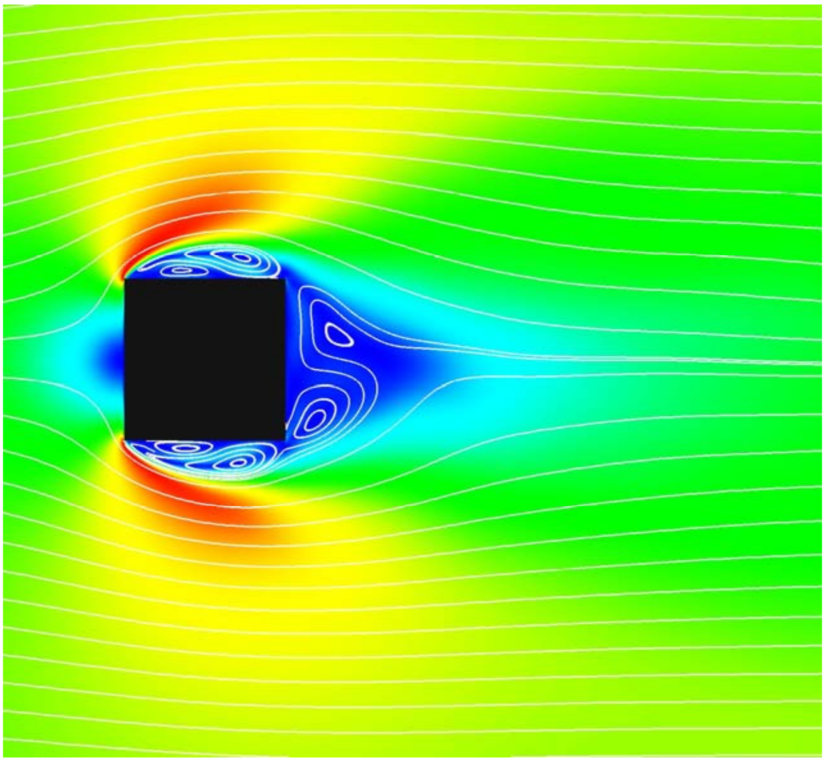

Figure 4. Time averaged velocity along the centreline.

the closest neighbourhood of the bypassed walls.

4. CONCLUDING REMARKS

This contribution has been aimed to give some information about the ILES utilization together with the immersed boundary method. Especially form the comparison showed on the Figure 3 it should be said that results seem to be acceptable and also the value of St that is a little bit lower than in the other laboratory experiments confirms that the chosen methodology is acceptable. We suppose that even better results it will be possible to reach when some appropriate wall model will be applied.

ACKNOWLEDGEMENTS

This research has been supported by the Grant Agency of the Czech Academy of Sciences, grant no. T400300414, by the Grant Agency of the Czech Republic, grant no. 205/06/0727 and by the research plan financed by the Czech Ministry of Education, Youth and Sports MSM0021620860.

REFERENCES

Boris, J.P, 1989, Whither Turbulence? Turbulence at the Crossroads. J.L. Lumley (ed.), Springer-Verlag, Berlin.

Bouris, D., Bergeles, G.,1999, 2D LES of vertex shedding from a square cylinder, J. Wind Eng. Ind. Aerodyn. 80, 31—46.

Breuer, M., Pourquie, M., 1996, First experiences with LES of flows past bluff bodies, in: W. Rodi, G. Bergeles (Ed.), Proc. 3rd Int. Symp. on Engineering Turbulence

Modelling and Measurements, Heraklion-Crete, Greece, 27—29 May 1996, Elsevier, Amsterdam, pp. 177—186.

Durao, D., Heitor, M., Pereira, J., 1988, Measurements of turbulent and periodic flows around a square cross-section cylinder, Exp. Fluids 6,298—304.

Kim, J., Kim, D. and Choi, H., 2001. An Immersed-Boundary Finite-Volume Method for Simulations of Flow in Complex Geometries. J. Comput. Phys. 171 132–150.

Kurganov, A and E. Tadmor, 2000, New High-Resolution Central Schemes for Nonlinear Conservation Laws and Convection-Diffusion Equations, J. Comput. Phys. 160, 241.

Lyn, D., Einav, S., Rodi, W., Park, J., 1995, A laser doppler velocimetry study of ensemble-averaged characteristics of the turbulent near wake of a square cylinder, J. Fluid Mech. 304, 285—319.

Murakami, S., Mochida, A., 1995, On turbulent vortex shedding flow past a square cylinder predicted by CFD, J. Wind Eng. Ind. Aerodyn. 54,191—211.

Peskin, C. S., 1982. The fluid dynamics of heart valves: Experimental, theoretical, and computational methods. Annu. Rev. Fluid Mech. 14 235–259.

van Leer, B., 1979. Towards the Ultimate Conservative Difference Schneme V. A Second-Order Sequel to Godunov’s Method. J. Comput. Phys. 32 101–136.