17th International Conference on
Harmonisation within Atmospheric Dispersion Modelling for Regulatory Purposes
9-12 May 2016, Budapest, Hungary

A HYBRID APPROACH FOR THE NUMERICAL SIMULATION OF FLOWS IN URBAN ENVIRONMENT

Bence Hermann and Miklós Balogh

Department of Fluid Mechanics, Budapest University of Technology and Economics, Budapest, Hungary

Abstract: Observing the nature of the air flows in urban environment is essential in the understanding of processes which have influence on urban climate and air quality. Numerical simulation is one way to achieve this. However, CFD simulations of atmospheric flows in these environments require huge amounts of computational capacity if the whole geometry is fully detailed. There are techniques to reduce this requirement, while still producing acceptable results. One of these is the hybrid method, which uses source terms in the transport equations to model the effects of the buildings implicitly. The geometry is explicitly modelled only around the target area of analysis. This method can drastically reduce the cell number, resulting in a much faster numerical simulation. The aim of this study is to create an OpenFOAM solver which is capable of handling this porosity drag force approach. The parameters of the source terms are calculated by performing a series of CFD calculations with different cuboid-shaped buildings using cyclic boundary conditions. A local drag coefficient can be obtained for each cell level between the ground and the top of the building, for each building geometry. This is the main parameter of the source terms. The final goal was to implement the hybrid model in OpenFOAM with the parametrized implicit approach, and it is compared to the results of explicit CFD simulations.

Key words: *porous drag-force approach, parametrization, RANS, OpenFOAM, height dependent drag coefficient*

INTRODUCTION

The Reynolds-Averaged Navier-Stokes (RANS) modelling of atmospheric boundary layer flows has always been a widely researched topic as it can assist in the solution of a broad range of engineering problems. In many cases, however, there are various types of roughness elements on the ground. Resolving these details with the mesh vastly increases the computational requirements, so the modelling of the effects of these elements on the flow is advisable. One way to do this is the porous drag-force approach, which utilizes sink terms in the transport equations to avoid explicit modelling of the geometry, reducing the computational cost. This approach has been used to model the effects of roughness elements such as vegetation, trees, crops, forests on the flow, investigated by e.g. Green (1992) and Liu et al. (1996). Yee, Lien (2004-2005), Balogh and Kristóf (2010) among others examined the implicit modelling of building arrays, which is the subject of the current investigation.

In this present study the parametrization of an OpenFOAM solver are performed utilizing a so-called hybrid method based on numerous CFD simulations with explicitly modelled geometries. This method combines the two ways by using the porous drag-force approach in the less important, peripheral regions and modelling the geometry explicitly at the target area of analysis.

NUMERICAL SIMULATIONS WITH EXPLICIT GEOMETRY

A series of RANS simulations had to be performed to obtain the necessary parameters for the porous drag force approach. In these we examined the vertical distribution of the field variables around a surface-mounted cuboid with cyclic (or in other words periodic) boundary conditions, which models one building of a building array as seen on Figure 1. The cuboids were described using the following two geometric parameters:

- Building density [-]: $\lambda = \dfrac{L_{dx} L_{dy}}{L_x L_y}$

- Building height [m]: H

We looked to obtain a local drag coefficient which depends on these two geometric parameters and the distance from the ground. The base cross section of the buildings was assumed to be a square.

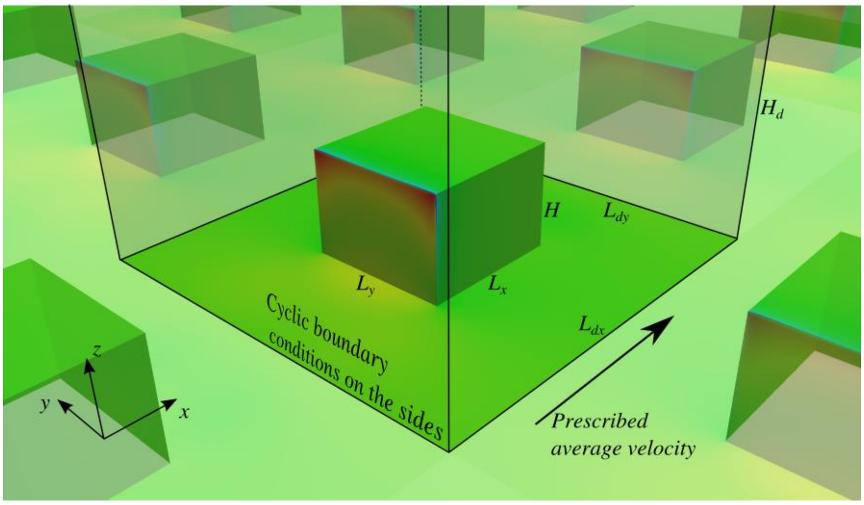


Figure 1. Computational domain and geometric dimensions

We used the blockMesh utility of OpenFOAM to create a structured mesh around the building model. To improve the solution quality and convergence, cells with high aspect ratios such as those near the solid surfaces could not be allowed to appear at the cyclic boundaries of the domain, as seen below on Figure 2.

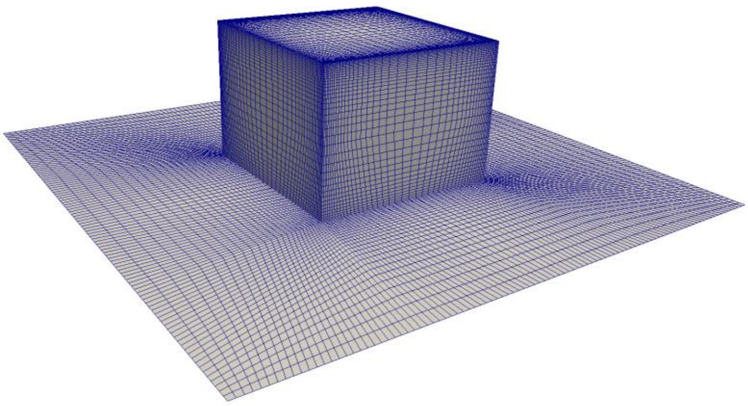


Figure 2. Surface mesh on the building walls and ground surfaces

The flow was assumed to be incompressible and steady-state, which greatly reduced computational time. Since the boundary conditions are cyclic on all sides of the domain, an explicit pressure gradient source term was implemented in the momentum equation to allow a steady-state solution. This source term was automatically adjusted to keep a prescribed average velocity constant in the domain, which is the initial condition for the calculations in all cells. The prescribed value is calculated by averaging the velocity profile introduced by Richards and Hoxey (1993) for 2D incompressible steady-state modelling of the atmospheric boundary layer using $k - \varepsilon$ turbulence model:

$$U(z) = \frac{u_\tau}{\kappa} \ln\left(\frac{z + z_0}{z_0} \right) \tag{1}$$

With a given reference velocity of U_{ref} = 3 ms^{-1} at the reference height of z_{ref} = 10 m and the known ground surface roughness z_0, the friction velocity could be calculated, fully defining the profile The direction of this prescribed velocity was parallel with the x-axis for our current investigation. The cuboid can be considered a bluff body, which means that the Reynolds number dependence of the drag coefficient should be negligible. Therefore the averaged velocity was kept constant throughout all calculations.

The turbulence model used in the simulations was a modified version of the standard $k - \varepsilon$ model, where a source term used by Parente and Benocci (2010) was added to the dissipation rate transport equation. At the ground and the building walls the rough wall functions proposed by Balogh (2014) were used. The k and ε values were also initialized using the profiles proposed by Richards and Hoxey (1993):

$$k(z) = \frac{u_\tau^2}{\sqrt{C_\mu}} \tag{2}$$

$$\varepsilon(z) = \frac{u_\tau^3}{\kappa(z + z_0)} \tag{3}$$

The numerical simulations took 4-5 days each on a single processor core, which is a very long time for a RANS simulation with a relatively small cell count (1-1.5 million cells). The reason for this was the cyclic boundaries, which vastly increased the number of iterations.

PARAMETRIZATION OF THE DISTRIBUTED DRAG FORCE APPROACH

The next step was to extract the vertical distribution of the field variables from the converged solutions, which was realized by averaging the values at the cyclic boundaries in each horizontal cell layer of the structured mesh. All field variables depend on λ, H, and the vertical ground distance. For our goals it was necessary to fit analytical expressions to the obtained curves, which was performed in three steps. First, an individual profile was fit in each case for the different λ, H pairs. Then, the λ dependence of the parameters of these expressions was described with second-order polynomials. Finally, the building height dependence of the parameters of these polynomials was approximated with another second-order expression. The final two steps are equivalent to fitting a second-order surface to each parameter of the individual expressions. These operations were performed with the built-in Levenberg-Marquardt nonlinear least-squares fitting algorithm in the optim package for Octave (“leasqr” function).

The velocity profile could be approximated with a sum of a tangent and a linear function below the building height, and a logarithmic function above it, which have six parameters altogether:

$$U(\tilde{z}) = \begin{cases} A_{U1} \cdot \tan\left(A_{U2}(\tilde{z} - A_{U3})\right) + A_{U4} + A_{U5}\tilde{z} = f_{U1}(\tilde{z}) \\ A_{U5} \cdot \ln(\tilde{z}) + f_{U1}(1) = f_{U2}(\tilde{z}) \end{cases} \qquad where\ \tilde{z} = \frac{z}{H} \tag{4}$$

The problem with this approximation is that it does not ensure the continuity of the derivative of $U(z)$. For this reason a blending function was introduced in the following way to avoid using conditional expressions:

$$U(\tilde{z}) = \left(1 - f_{blend}(\tilde{z})\right) \cdot f_{U1}(\tilde{z}) + f_{blend}(\tilde{z}) \cdot f_{U2}(\tilde{z}) \quad , where \quad f_{blend}(\tilde{z}) = \frac{\tanh(B \cdot (\tilde{z} - 1)) + 1}{2} \tag{5}$$

This is basically a ’’derivable’’ Heaviside step function if the B parameter is a high value. It is also important to note that the points at the very top of the domain were neglected for the curve fitting, because the zero gradient boundary condition "straightened" out the logarithmic velocity profile, which was unphysical.

The turbulent kinetic energy and its dissipation were approximated with the two expressions below:

$$k(\tilde{z}) = A_{k1} e^{-A_{k2}(\tilde{z}-1)} - A_{k3} e^{A_{k4}(-\tilde{z}-A_{k5})} \tag{6}$$

$$\varepsilon(\tilde{z}) = \left(1 - f_{blend}(\tilde{z})\right) \cdot \left(\frac{A_{\varepsilon 1}}{\tilde{z}} - \frac{A_{\varepsilon 2}}{\tilde{z}-1} + A_{\varepsilon 3}\tilde{z} + A_{\varepsilon 4} \right) + f_{blend}(\tilde{z}) \cdot \frac{A_{\varepsilon 5}}{\tilde{z} - A_{\varepsilon 6}} \tag{7}$$

For each $A_{fv,j}$ (the j-th parameter of the field variable *fv*) a two-step surface fitting was performed. First, for each *H*, the λ dependence of the $A_{fv,j}$ parameters was estimated with a second-order polynomial, then the building height dependence of the $A_{\lambda i}$ parameters was approximated with a quadratic expression as well:

$$A_{fv,j} = A_{\lambda 1} \cdot \lambda^2 + A_{\lambda 2} \cdot \lambda + A_{\lambda 3} \tag{8}$$

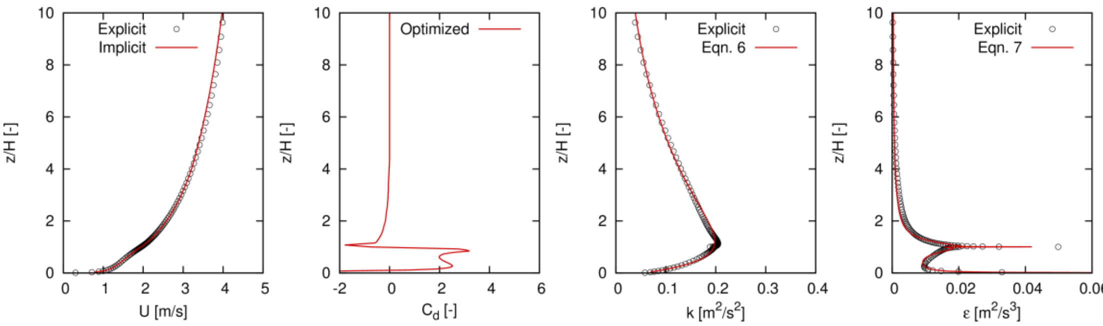
$$A_{\lambda,j} = A_{H1} \cdot H^2 + A_{H2} \cdot H + A_{H3} \tag{9}$$

An example for function parameter fitting can be seen on Figure 3 for $H = 6.5$ meter and $\lambda = 0.075$. The empty circles are the values from the explicit CFD simulations, while the continuous red curves are the fitted analytical expressions.

Based on the results of the simulations with the explicit approach, we could move on to the programming of the OpenFOAM solver. To implement the building resistance implicitly into the solver, we added the following source term into the momentum equation in the desired regions:

$$S_i(z) = \frac{\rho}{2} C_D(z) A_f U(z) u_i(z) \tag{10}$$

A_f is the cross-section area of the building per unit volume, *U* is the velocity magnitude and u_i is the velocity component in the *i*-th direction. The vertical distribution of the drag coefficient was obtained by a solver written for this purpose. We ran simulations with a 2D domain with cyclic boundary conditions, basically a slice of the domain seen on Figure 1 in the x-z plane without the building. The value of C_d at height *z* was continuously modified throughout the iterations according to the difference between the actual and desired velocity distributions, which were calculated according to equation (5).



Figures 3. Vertical profiles for velocity, turbulent properties and sectional drag coefficient.

RESULTS AND DISCUSSION

On Figure 3 it can be observed that the velocity distribution results with the implicit method provide a good approximation of the explicit case, and the necessary local drag coefficient distribution to obtain this can be seen next to it. It can be seen that in order to precisely achieve the desired vertical velocity distribution it is not enough to utilize the source term in the region below the building height, but a value different from zero should be prescribed above it. Simply calculating the sectional drag coefficients from the wall forces would give a significant error mainly because of this.

The implemented hybrid solver is able to handle arbitrary amount of implicit and explicit regions, making it viable for numerous urban atmospheric CFD problems. This mainly includes providing realistic boundary conditions for the target area of analysis, e.g. calculating the forces acting on a certain building, pollutant dispersion in urban areas or planning of the location of wind turbines.

SUMMARY AND OUTLOOK

A novel hybrid solver is developed for the simulation of urban flows and it is parameterized based on a series of explicit CFD simulations. Additionally, analytical profiles are fitted for U, k and ε based on the explicit results. These profiles could serve as boundary conditions in CFD simulations, where the domain is limited to a bounded region of a city. The implicit parametrization scheme of the hybrid method ensures good agreement between the explicit and implicit volume fluxes with the height dependent drag coefficient values imposed in each cell.

In order to extend the generality of the parametrization, varying building side ratios should be investigated with various flow directions. The implementation of source terms for the turbulent kinetic energy and its dissipation is also planned to ensure pronounced matching between the explicit and implicit turbulent properties. These source terms can be calculated from the analytical profiles in equations (6) and (7) and the previously obtained *Cd* profiles, and implemented similarly to Green et al. (1995) and Balogh and Kristóf (2010). The solver can then be validated against the MUST (Mock Urban Setting Trial) experimental data, similarly to Balogh (2014).

ACKNOWLEDGEMENTS

The project presented in this article is supported by the projects K 108936 "Flow and dispersion phenomena in urban environment" of the Hungarian Scientific Research Fund and the New Széchenyi Plan project TÁMOP-4.2.1/B-09/1/KMR-2010-0002 "Development of quality-oriented and harmonized R+D+I strategy and functional model at BME".

REFERENCES

P.J. Richards and R.P. Hoxey, 1993: Appropriate boundary conditions for computational wind engineering models using the k-ε turbulence model. *Journal of Wind Engineering and Industrial Aerodynamics*, **46 & 47**, 145-153.

E. Yee and F. Lien, 2004: Numerical Modelling of the Turbulent Flow Developing Within and Over a 3D Building Array, Part I: A High Resolution Reynolds-averaged Navier-Stokes Approach. *Boundary-Layer Meteorology*,**112,** 427-466.

E. Yee and F. Lien, 2005: Numerical Modelling of the Turbulent Flow Developing Within and Over a 3D Building Array, Part II: A Mathematical Foundation for a Distributed Drag Force Approach. *Boundary-Layer Meteorology*,**114,** 245-285.

E. Yee and F. Lien, 2005: Numerical Modelling of the Turbulent Flow Developing Within and Over a 3D Building Array, Part III: A Distributed Drag Force Approach, Its Implementation and Application. *Boundary-Layer Meteorology*,**114,** 287-313.

E. Yee and F. Lien, 2004: A Distributed Drag Force Approach for the Numerical Simulation of Urban Flows. *DRDC Suffield TR 2004-169*

M. Balogh, 2014: Numerical simulation of atmospheric flows using general purpose CFD solvers. *PhD thesis*

M. Balogh and G. Kristóf, 2010: Fine scale simulation of turbulent flows in urban canopy layers. *Időjárás, Quarterly Journal of the Hungarian Meteorological Service,* **114,** 135-148.

S.R. Green, 1992: Modelling turbulent air flow in a stand of widely-spaced trees. *Journal of Computational Fluid Dynamics and Its Applications*, **23,** 294-312.

S.R. Green, J. Grace, and N.J. Hutchings, 1995: Observations of turbulent air flow in three stands of widely spaced sitka spruce. *Agricultural and Forest Meteorology*, **74,** 205-225.

J. Liu, J.M. Chen, T.A. Black, and M.D. Novak, 1996: E-ε Modelling of Turbulent Air Flow Downwind of a Model Forest Edge. *Boundary-Layer Meteorology*, **77,** 21-44.

A. Parente and C. Benocci, 2010: On the RANS simulation of neutral ABL flows. *The Fifth International Symposium on Computational Wind Engineering (CWE2010)*