Abstract — The analysis of turbulent flows in urban areas is part of our research for simulating meso- and microscale atmospheric flows, since it is of great significance with respect to civil design and environmental protection. The application of general purpose computational fluid dynamics (CFD) solvers for the examination of smaller scale atmospheric phenomena, such as urban flows, has many advantages due to their flexible use on arbitrary meshes with complex geometries. A CFD solver has been adapted to atmospheric applications by using a purpose developed transformation method along with some volume sources active in the transport equations. In this way, the effects of the thermal stratification, adiabatic temperature variation, compressibility, and the Coriolis force were taken into account. Our present work is aimed at the development of a simulation methodology and software components for solving urban ventilation problems with special respect to the modeling of urban canopy layers. An advantage of the CFD based modeling technique is that the mesh size could change continuously in the computational domain according to the location of the building arrays, the road-system, and the examination area. This paper presents the modeling concept and its functionality in practice.

Key-words: CFD, urban canopy, turbulent flows, distributed drag force approach, two-equation model

1. Introduction

The detailed description of turbulence in the atmospheric boundary layer (ABL) is essential with regard to the dispersion and heat transfer processes, both within and above urban canopies. The realization of the accurate description is very difficult due to the complex structure of an urban canopy layer, which is a varied
system of bluff obstacles including trees and buildings. The examination of the urban climate, including the ventilation of the city, is possible by using statistical methods, such as roughness parameter mapping (Gál and Unger, 2009) or using dynamical methods. Though the detailed numerical simulation of flows in an urban canopy layer is possible by using CFD techniques, the higher numerical cost of the spatial discretization of the complex geometries makes this unrealizable in practice for very large domains. Finding the balance between the numerical cost and reasonable results is very important with regard to the civic design and environmental protection.

A potential technique, which keeps the balance, and furthermore, conforms to the variation of surface coverage, could be a hybrid method, such as a combination of the explicit and implicit description of the flow properties in an urban canopy. For simulating turbulent flows in the mostly exposed areas (e.g., some parts of the downtown area) where we need the most detailed results, the explicit modeling of the buildings could be used, and in the other parts of the examination area, the distributed drag force approach (Green, 1992; Liu et al., 1996), as an implicit technique, could be applied with much lower numerical costs, although resulting in a lower resolution.

The efficient modeling of the flows in the atmospheric boundary layer, including turbulence, is feasible by solving the unsteady or steady state Reynolds-Averaged Navier-Stokes (URANS and RANS) equation due to the relatively low computational cost and reasonable accuracy. One of the most common turbulence model used in microscale investigations is the $k-\varepsilon$ two-equation model both for unsteady (URANS) and steady (RANS) simulations (Hargreaves and Wright, 2007). The main goal of this study is to develop an efficient hybrid method for simulating turbulent urban flows with CFD techniques.

2. Application of a general purpose CFD solver for atmospheric simulations

The use of a general purpose CFD solver adapted to atmospheric applications (Kristóf et al., 2009) based on the realizable $k-\varepsilon$ model (Shih et al., 1995) could be efficient for simulating urban ventilation and heat island problems in practice. The most common solvers are not capable of handling all of the physical processes in the arbitrarily stratified atmosphere, hence a mathematical transformation has been developed (Kristóf et al., 2009). These are implemented in the ANSYS-FLUENT simulation system as a user defined function package, and validated with well documented laboratory experiments, analytical solutions, and widely accepted numerical results. This function package has been extended with functions for taking into account other important physical effects in connection with geophysical use, such as Coriolis force effects validated by analytical solutions. The results of the simulations of non-
hydrostatic phenomena, such as a descending cold air bubble, laboratory scale gravity waves, as well as urban heat island circulation agree well with references. The implemented physical processes, along with the distributed drag force approach, allow for the simulation of non-hydrostatic atmospheric flows in an urban environment. The present study focuses on modeling the impact of the obstacle arrays in the urban canopy layer, although, the examination of heat island phenomena is also in progress.

3. Distributed drag force approach

Recently, the application of CFD techniques is increasing in the field of micrometeorology, thus the knowledge of the flow structure and turbulence in an urban canopy layer is rapidly developing as well. Many scientists are working on the development of a distributed drag force approach, both for vegetated canopies (e.g., Green, 1992; Liu et al., 1996) and for building arrays (Lien and Yee, 2004, 2005; Lien et al., 2004, 2005; Carissimo and MacDonald, 2002). Therefore, it already has a solid theoretical background. This approach has already been used in practice for the urbanization of weather prediction models (Hamdi and Masson, 2008) to implicitly take into account the effect of buildings.

The essence of the drag force approach is an additional drag term in the momentum equation and two other terms in the transport equation of the turbulent kinetic energy \( k \) and the turbulent dissipation rate \( \varepsilon \). The drag term of the momentum equation is composed of the viscous and the form drags, while the value of the viscous component is much lower than the form component. Therefore, the former could be neglected. With this simplification, the source term of the momentum equation has a general form of

\[
S_i = - \rho C_d A_i U u_i ,
\]

where \( \rho \) is the air density, \( C_d \) is the drag coefficient, \( A_i \) is the frontal area per unit volume normal to the \( i \)th direction, \( U \) is the velocity magnitude, and \( u_i \) is the velocity component in the \( i \)th direction. The unit of the momentum source is \( N \text{ m}^{-3} \text{ s}^{-1} \).

In the case of vegetated canopies, the obstacles (e.g., branches and leaves) convert the kinetic energy of the flow into wake turbulences with a smaller length scale than the shear-generated turbulence. Therefore, the canopy yields a net turbulent kinetic energy loss (Green et al., 1995) instead of enhancing the wake production. This could be modeled with a source term in the following form,

\[
S_k = \rho C_d A_f \left[ \beta_p U^3 - \beta_d U k \right] ,
\]
where \( A_f \) is the total frontal area per unit volume, \( k \) is the turbulent kinetic energy, \( \beta_p \) constant is a fraction of the mean flow kinetic energy produce \( k \), and \( \beta_d \) is an empirical constant for short-circuiting the turbulent cascade (Green, 1992). The unit of the turbulent kinetic energy source is kg m\(^{-3}\) s\(^{-1}\).

The simplest model of the turbulent dissipation rate source term is based on the Kolgomorov’s relation, which yields

\[
S_\varepsilon = C_{\varepsilon 4} \frac{\varepsilon}{k} S_k . \tag{3}
\]

The units for the turbulent dissipation rate source are kg m\(^{-4}\) s\(^{-1}\). In Eq. (3), \( \varepsilon \) is the turbulent dissipation rate and \( C_{\varepsilon 4} \) is a constant. This relation was improved by Liu et al. (1996) providing a better fit to wind tunnel data. Accordingly, an alternative model could be defined as a more general form, which reads

\[
S_\varepsilon = \rho C_d A_f \left[ C_{\varepsilon 4} \beta_p \frac{\varepsilon}{k} U^3 - C_{\varepsilon 5} \beta_d U \varepsilon \right] . \tag{4}
\]

In Eq. (4), the new constant \( C_{\varepsilon 5} \) defines the mixing length anisotropy, if it is not equal to \( C_{\varepsilon 4} \). Otherwise, the alternative model of Eq. (4) turns into the simpler Eq. (3) as noticed by Sanz (2003). It should be mentioned, that the source term of the turbulent dissipation rate is required in those microscale simulations that are based on two-equation turbulence models. In mesoscale models these terms can be neglected (Otte et al., 2004). The coefficients in Eqs. (2)–(4) depend on the type of the turbulence model applied and the characteristics of the canopy layer. The relations between these constants and the constants of the \( k-\varepsilon \) model, together with the characteristics of the vegetated canopy were suggested by Sanz (2003) and were analyzed by Katul et al. (2004) and Sanz and Katul (2007).

In the present studies, the source terms of the momentum, turbulent kinetic energy, and turbulent dissipation rate were modeled in the above presented forms, applying the alternative model for describing \( S_\varepsilon \). In vegetated areas, the properties of the drag terms were based on the works of Balczó et al. (2009), while in building arrays, the drag coefficient \( C_d \) was calculated as a function based on the volumetric porosity of the obstacle arrays (Appendix A) applied by Coirier and Kim (2006) in a similar application.

The goal of our method is to provide a mesh which follows the outlines of open areas, such as streets, squares, and open fields. The drag source terms should not be applied in these areas if the local mesh resolution is sufficiently high, thus explicit modeling is possible. Therefore, the impacts of these
cavities can be taken into consideration, although this strongly depends on mesh resolution.

4. **An example on modeling turbulent flows in an urban canopy layer**

In this section, we introduce a typical and practical application of CFD techniques in modeling turbulent atmospheric flows in urban areas. In this case study, the examination area is the 11th district of Budapest (the capital of Hungary), where a diversified landscape could be found, ranging from rural to downtown areas. Since this region is also surrounded by diversified regions, the examination area was extended with a relaxation zone for simplifying the setup of the lateral boundary conditions (Appendix B). In this way, the dimensions of the examination area in the x, y, and z directions were 9155, 7150, and 1800 m, respectively, while the computational domain extended by the relaxation zone (Fig. 1) is twice the original.

![Fig. 1. Top view of the computational domain colored by the elevation. The similar surface coverage regions are highlighted (fine solid lines) in the examination area (thick dashed line), which is extended with the relaxation zone (thick solid line). The circles denote the sampling points of different canopy profiles used later.](image)

The geometrical setup of the computational domain was based on the SRTM (Shuttle Radar Topography Mission) elevation database and a raster graphical map. It contains surface coverage and building cluster data in a simplified, type dependent form. The elevation data could be used after a coordinate transformation from WGS84 to a Cartesian frame of reference, namely the Uniform National Projection system (Hungarian abbreviation:
Nevertheless, the surface coverage data needed a pre-processing procedure for efficient use in modeling. As a result of these, the properties of the canopy layer such as the canopy layer height, the frontal area and solid fraction of the obstacles are available at each region of the computational domain. Since the domain contains several regions, we only introduce some typical parameters of urban type canopies (Table 1).

Table 1. Typical values of the total canopy height $H$, the solid fraction $\lambda$, and the frontal area per unit volume $A_f$ in the urban type canopy, from different parts of the examination area.

<table>
<thead>
<tr>
<th>Region</th>
<th>$H$ [m]</th>
<th>$\lambda$ [-]</th>
<th>$A_f$ [m$^2$ m$^{-3}$]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Suburban (sample C)</td>
<td>7</td>
<td>0.3</td>
<td>0.55</td>
</tr>
<tr>
<td>Block of flats (sample B)</td>
<td>20</td>
<td>0.4</td>
<td>0.63</td>
</tr>
<tr>
<td>Downtown (sample A)</td>
<td>15</td>
<td>0.6</td>
<td>0.77</td>
</tr>
</tbody>
</table>

Due to the finite volume method used in our simulation system, the spatial discretization of the domain, based on the computational mesh, is a system of wedge cells. This mesh was generated from the elevation data and the polygons are bounding the areas with the same type of surface coverage. The cells are triangular wedges with vertical orientation and their lateral edge length is varying between 8 and 160 meters in the examination area, growing up to 1000 meters in the relaxation zone, with a cell growth rate 1.3. For open fields, such as wider streets, parks, and the Danube River, the mesh size is the minimum possible. Therefore, the impact of these could be described at a higher resolution, as it can be seen in Fig. 2.

Vertically, the cells are ordered in layers and their height is increasing from 3 to 530 meters with the distance from the ground surface. The layers follow the terrain near the ground and become flat when approaching the upper boundary.

At the inlet boundaries the velocity components, turbulent kinetic energy, and dissipation rate were defined as pre-calculated profiles shown in Fig. 3. These were calculated by a one-dimensional, steady-state, realizable $k-\varepsilon$ model with the reference surface coverage conditions.

The wind climate in the district could be examined by averaging the results of the simulations executed for different boundary conditions. In the present example, eight runs were performed with different boundary conditions for the primary wind directions having the same velocity magnitude at 10 meters, which was 3 m s$^{-1}$. The simulations were steady state runs, while both the non-hydrostatic and Coriolis force effects were neglected. Therefore, only the source terms of the porous drag model and the non-reflective diffusion were enabled. The solver was used with second order upwind schemes for the spatial...
discretization of the momentum equation, the turbulent kinetic energy and
turbulent dissipation rate transport equations, and a second order scheme for the
pressure equation. For better numerical stability, the SIMPLE pressure-velocity
coupling was used, along with the node based Green-Gauss gradient scheme.
With these settings, the simulations for the eight wind directions took only six
hours using a Quad-Core computer.

Fig. 2. The structure of the computational mesh of the downtown area of the 11th
district, where the shaded areas show the building arrays, and the white ones denote
uncovered areas such as streets, squares, and open fields, together with vegetated areas.

Fig. 3. Vertical profiles at the inlet boundaries, where the velocity magnitude (black) is
scaled by the reference value at 10 meters above the ground. The turbulent kinetic energy
(dashed) and dissipation rate (dashed-dotted) are normalized by the values next to the wall.
5. Results

Vertical profiles of the weighted average of the velocity magnitude and the turbulent shear stress, plotted along lines selected from different canopy regions from sampling points A, B, and C in Fig. 1, are verified using analytical canopy profiles. The analytical canopy profiles were published by Finnigan and Belcher (2004), calculating the velocity as

\[
U(z) = \begin{cases} 
\frac{u_T}{\kappa} \ln \left( \frac{z-H+d}{z_0} \right), & \text{if } z > H, \\
U_H e^{\beta(z-H)/l_m}, & \text{if } z \leq H,
\end{cases}
\]

and the shear stress as

\[
\tau(z) = l_m^2 \left[ \frac{dU}{dz} \right]^2,
\]

where \(U(z)\) and \(\tau(z)\) are the velocity and the shear stress magnitudes, \(u_T\) is the friction velocity, \(\kappa\) is the von Kármán constant, \(z\) is the height above the ground, \(H\) is the height of the canopy, \(U_H\) is the velocity magnitude at the top of the canopy, \(d\) is the displacement height, \(z_0\) is the roughness height, \(l_m\) is the mixing length, and \(\beta\) is a constant of the profile. Note that \(u_T\), \(z_0\), \(d\), and \(l_m\) are functions of the canopy density. These fall within the range of the analytical results, for low and high canopy densities (Fig. 4), although some differences could be found in the shape of the profiles farther on the ground surface. The reason of these differences could be that the analytical profiles were calculated for flat surface, while our simulations were applied on complex terrain.

\[\text{Fig. 4. Normalized velocity and shear stress profiles as a function of the height scale } z/H, \text{ where } U_H \text{ is the velocity at the top of the canopy and } \tau' = \rho u_T^2. \text{ Sampling points A, B, and C can be seen in Fig. 1.}\]
The averaged flow fields, which characterize the different regions of the district concerning the ventilation, are calculated from the results of the different cases weighted by the probability of the case dependent wind direction, both for the velocity magnitude and turbulent quantities. The average velocity magnitude as a quantitative parameter predicts the ventilation of the different regions, while the turbulent intensity contains useful information about the turbulent fluctuations. These were plotted at different heights above the ground, namely at 10 and 30 meters. The velocity and turbulent intensity fields are also scaled by the reference values of those calculated from the inlet profiles taking into consideration the local elevation. Since the inlet profiles are defined for an undisturbed free flow over a smooth surface, the scaled fields express the impact of the topography of the examination area and the topology of the canopy layer.

![Fig. 5. Distribution of the velocity magnitudes scaled by reference values (inlet profiles) in the examination area, at 10 meters above the ground.](image)

Near the surface, as shown in Fig. 5, over open areas, such as streets, squares, parks, and over the Danube River, the velocity magnitudes are significantly higher than the reference values due to the horizontal displacement of dense regions where the flow is moderated by obstacles. Moving away from the surface, the velocity is increasing as an effect of the canopy reducing blocking, although above the canopy layer height, the impact of the drag is still realizable (Fig. 6).

The scaled turbulence intensity has a local minimum near the surface (Fig. 7), since the reference turbulence intensity has the maximum next to the wall. The reference profiles were calculated with free flow conditions, thus, only
the wall has an impact on its turbulent properties. At 30 meters above the ground, shown in Fig. 8, the turbulence develops at the boundaries of the blocks, mainly where the canopy properties change suddenly. This effect is stronger on those side of the canopy blocks where the gradient of the porous drag has a high positive value in the streamwise direction of the locally dominant wind.

Fig. 6. Distribution of the velocity magnitudes scaled by reference values (inlet profiles) in the examination area, at 30 meters above the ground.

Fig. 7. Distribution of the turbulent intensity scaled by reference values (inlet profiles) in the examination area, at 10 meters above the ground.
6. Conclusions and further developments

After the implementation of the distributed drag force parameterization, a practical application was executed, which demonstrates the capabilities of the CFD based approach in fields of the urban climatology and pollution control. The source term of the parameterization was applied within the areas where obstacles were found, thus, the impacts of the street canyons could also be considered. The properties of the canopy layer were also changed according to their type. The averaged results of the simulation were verified with analytical canopy profiles in representative points and good qualitative agreement has been found.

The results of the current study are useful for the further development in modeling stratified canopy layers, including the effects of the heat island phenomena and thermal convection. The realization of this requires the adaptation of the parameterization schemes modeling heat transfer and storage in the urban canopy layer (e.g., Vu et al., 2002) with higher resolution.

Acknowledgements — This research was supported by the Hungarian Scientific Research Fund (Hungarian abbreviation: OTKA, grant No. T049573), and the wind climate examination supported by the Local Government of the 11th district of Budapest. The author would like to express special thanks for the data and information regarding the district, given free run by the Local Government.
Appendix A: Calculation of the drag coefficient

For the simplification of the drag coefficient \( C_d \), it was calculated as a function of the volumetric porosity of the obstacle array applied by Coirier and Kim (2006) in a similar application. After adapting this function to our model, the drag coefficient could be defined in the following way

\[
C_d = \begin{cases} 
\frac{\min(\Delta z, H - z_c + 0.5 \Delta z)}{[1 - \lambda_t] \Delta z}, & \text{if } z_c - 0.5 \Delta z \leq H \\
0, & \text{if } z_c - 0.5 \Delta z > H
\end{cases},
\]

(A1)

where \( \lambda_t \) is the total solid volume per unit volume composed by the volume of the buildings and the vegetation, \( H \) is the canopy height, \( z_c \) is the height of the cell centroid above the ground, and \( \Delta z \) is the height of the cell.

Appendix B: Simplified specification of the boundary conditions

Both the elevation and surface coverage data were relaxed in the space to their reference values along the relaxation zone from the edge of the examination area to the lateral boundaries with the use of a dumping function Eq. (6). The reference value for the elevation was its spatial average at the lateral sides of the domain, while for the specification of the relaxed surface coverage parameters, the properties of the open grassland were used as a reference. For this reason, identical vertical profiles could be defined at every inlet boundary, calculated with the reference surface coverage properties.

\[
\sigma(r) = \frac{1 + \cos r}{2}, \quad r \in [0, \pi].
\]

(B1)

The dumping coefficient \( \sigma(r) \) is a function of the normalized distance from the closest lateral boundary \( r \). In the relaxation zone, the elevation and the characteristics of the canopy continuously approach a reference value defined on the boundary by Eq. (7).

\[
\phi(r) = \sigma(r) \phi_{ref} + [1 - \sigma(r)] \phi,
\]

(B2)

where \( \phi \) and \( \phi_{ref} \) are the values of the relaxed parameter at the nearest part of the examination area and at the lateral boundary, respectively. The realization of the non-reflective boundary conditions could also be obtained by using
additional source terms (Bodony, 2006) in the relaxation zone as an analogy of Eq. (7), which is, written in a general form, is

\[ S_r (r, \phi) = -\frac{\sigma(r)}{\Delta t} \left[ \rho \phi - \rho_{\text{ref}} \phi_{\text{ref}} \right] , \]  

(B3)

where \( S_r \) is the non-reflective diffusion source term, \( \Delta t \) is the time-step size, \( \rho \) and \( \phi \) are the current, \( \rho_{\text{ref}} \) and \( \phi_{\text{ref}} \) are the reference values of the fluid density and the field variable of the transport equation, respectively. The field variable is the velocity in the source term of the momentum equation, the enthalpy in the energy equation, and \( \phi \) equals 1 in the continuity equation. Note that in steady simulations, the value of the time-step size could be replaced by a time scale, while in incompressible cases \( \rho \) is equal to \( \rho_{\text{ref}} \).

References


Gál, T., Unger, J., 2009: Detection of ventilation path using high-resolution roughness parameter mapping in a large urban area. Build Environ 44(1), 198-206


