Use of detection of coherent flow structures for better understanding of 3D flow fields in urban environment

Tamás Régert, Ph.D., István Goricsán, Ph.D., Márton Balczó, Károly Czáder, Prof. Tamás Lajos
Department of Fluid Mechanics
Budapest University of Technology and Economics
regert@ara.bme.hu

Abstract - The rapid development of flow measurement (particularly PIV) and numerical simulation results in a huge amount of information: 2D or 3D velocity, pressure and turbulent kinetic energy distribution of high resolution. Particularly in case of 3D distribution it is not at all easy to "understand" the flow field. Without clear understanding of the flow field characteristics it is difficult to develop proposals addressing e.g. improvement of a given air pollution situation. This paper discusses the use of advanced flow field structure extraction methods enabling a comprehensive understanding of the flow field structure presented on cases of parts of urban environment like streets, square, intersection of streets, as well as on simplified models of built environment. The turbulent flow is measured or modeled via RANS approach solved by means of Finite Volume Method using the commercial software FLUENT 6 or MISKAM 5. Post-processing of the results is based on simultaneous observation and analyses of streamlines, wall streak lines, vortex cores, iso-surfaces of the second invariant of the velocity gradient tensor (usually denoted by Q) and iso-surfaces of the total pressure (in case of CFD). Applying these methods, one can not only show the structure of separated flow regions but can determine which of these structures are predominantly influential in terms of development of aerodynamic characteristics and transport processes of the flow in urban areas. The gathered additional information may extend the support of city and traffic planning in making conceptual or optimization decisions.

Key words – Post-processing, CFD, coherent-structures in flow

Introduction

The dispersion and transport of pollutants in urban environment is a task of high significance in fluid dynamics research. The flow past buildings can be characterized by very high Reynolds number. The critical point of the investigation of this type of flow is the lack of accurate reference data on the real phenomenon due to the reliability problems of field tests (Schafer et al. 2005). The first phenomenon that is usually, however not always (see f.e.g. Uehara et al. 2000) neglected is the thermal stratification. The constant density investigations restrict only to cases when atmospheric wind is dominant. Due to the very complex geometry of a city, mostly wind tunnels are the most appropriate tools for obtaining answers on the required pollution related questions (Plate, E.J. (1982), VDI Guideline 3783 Part 12 (2000)).

The other, popular and very effective investigation tool is Computational Fluid Dynamics (CFD) that gains more and more significance on this field. Due to the high Reynolds number the flow is usually modeled by the concept of Reynolds-Averaged Navier-Stokes equations that nessecitates the application of turbulence models to compute the values of the Reynolds stresses (Franke et al. 2004). Robust turbulence models, like the widely used k-ε model were developed for high Reynolds number flows that fit well for various applications.

However, the regions of the flow field which are in the vicinity of walls or in separation bubbles and vortices can be characterized by conditions that are not properly modeled by these turbulence models. Even more problematic is the modeling of fine details like trees, windows, balconies that influence the generation or dissipation of turbulence and deflect the flow. These problems might introduce discrepancies in the characteristics of pollutant propagation, extension and diameter of vortices, even regarding their existence at all (Franke et al. 2004, Gromke and Ruck, 2007).

The following main area of the investigation is post-processing of the results obtained either via experiments, or via CFD. In most cases tracer gas is introduced into the flow to play the role of pollution and then the air is sampled at certain locations in the area of the city and magnitude of local immission will be determined. Although the resultant pollution load in the locations of interest is known, the reason why the given immission values were obtained, i.e. the flow structures resulting the given transport remain unknown.

Several works are nowadays focusing on the determination of the flow field both by means of experiments (e.g. Dezső et al. 2003) and CFD (Li et al. 2006). In this way not only the resultant loads but the whole flow mechanism will be available that can establish the right method for controlling the transport characteristics. This paper discusses post-processing methods that are well established at other fields of fluid dynamics but may be useful for the interpretation of very complex three dimensional flow fields in urban canopy. These methods can be applied for both steady and unsteady computations, but in this paper only steady computational results are shown.

1 Investigation of coherent structures on the streets

It is well known from several works on the field of urban flow modeling that vortices develop in street canyons, influencing significantly both the dispersion of pollutants and wind comfort conditions. The concept of coherent structures has been proposed by Hussain 1986 to describe the phenomenon of turbulence on a deterministic but chaotical way besides the widely applied stochastic, statistics-based approach. Nowadays coherent structure approach is the most powerful tool for turbulence research (Hussain 1986, Adrian 2007, Mathur et al. 2007). It is known that this approach was originally applied for unsteady flows as the definition of coherent structures is expressed in terms of coherence in time (Hussain 1986), not in space. The 'classical' coherent structures are vortices [Jeong and Hussain 1995, Hussain 1986) that play crucial role in the mechanism of turbulence, but alternative proposals are also present, like the most recently introduced concept of repellor and attractor curves by Mathur et al. 2007.

In the present case authors adopt coherent structure detection techniques from the turbulence research field and apply them on large scale steady computational results, but it can be also applied for results from measurements. Here authors would like to note that the flow structures extracted from a steady flow field by means of coherent structure extraction techniques are not coherent structures strictly in the original meaning. This way regions characterized by higher vorticity as opposed to deformation can be extracted. Similar approach was applied by Kolar 2006 for jets in a cross flow and by Lohász et al. 2006 for the results of Large Eddy Simulation over a bluff body.

In this paper authors apply techniques for extraction of coherent structures on steady flow fields over a portion of Budapest city centre and over a simplified matrix of buildings. The flow field was obtained by numerical simulation using software Miskam and Fluent 6. Beside the well known but mostly confusing streamline patterns the authors present here more interpretable skeletons of complicated flow fields. The results of computations were post-processed by software Tecplot 360.

1.1 Application of the second invariant of the velocity gradient tensor (Q)

The "skeleton" builds up from the iso-surfaces of the second invariant of the velocity gradient tensor which property is usually denoted by Q (Jeong and Hussain 1995). The Q property is defined in Eq. (1) and interpreted as follows: Q expresses the domination of swirling flow as opposed to the deformation of the fluid particles.

$$Q = \frac{1}{2}\left(tr(\mathbf{\Omega}\cdot\mathbf{\Omega}^T) - tr(\mathbf{S}\cdot\mathbf{S}^T)\right) = -\frac{1}{2}\partial_i u_j \cdot \partial_j u_i \qquad (1)$$

where $\mathbf{\Omega}$ is the anti-symmetric part and **S** is the symmetric part of the velocity gradient tensor, u_i and u_j are the mean velocity components (i, j = 1, 2, 3; where 1 = x, 2 = y, 3 = z in traditional frame of reference notation). As vorticity induces potential-like vortex flow in the environment of the vortex filament due to the Biot-Savart law, the Q quantity thus detects structures in the flow field that are of high dynamical significance on their environment. This can be expressed as follows: if these structures are modified then large scale modification in the surrounding area can be achieved.

1.2 Application of the total-pressure based method

In case of flow with rotating fluid particles the total pressure is dependent on the integral value of the rotation integrated along the radius of the vortex (mathematically represented by the 'rotation' or 'curl' operation) (Eq.(2)).

$$\Delta p_{tot} = \rho \int_{r_1}^{r_2} \left(\bar{u} \times rot(\bar{u}) - \nu rotrot(\bar{u})\right) d\bar{r} \qquad (2)$$

where ρ[kg/m^3] is the density of the air, ν[m^2/s] is the kinematic viscosity of the air. It can be shown that the second term under the integral in Eq. (2) is negative in a cross section of a practically occurring vortex, the value of the difference in total pressure is increasing when the value of the rotation increases, i.e. the total pressure is decreasing at approaching the axis of the vortex. The drawback of the total-pressure based coherent structure extraction method is that its value decreases strongly not only in vortices but also in shear layers and boundary layers where the value of rotation is high but also the deformation of fluid particles is significant.

1.3 Analysis based on wall streak-lines

Wall streak-lines show lines the tangent of which are the shear force vectors. Separation and reattachment regions can be detected as well, as regions characterized by large shear stress that influences the wind comfort conditions.

1.4 Vortex core analysis

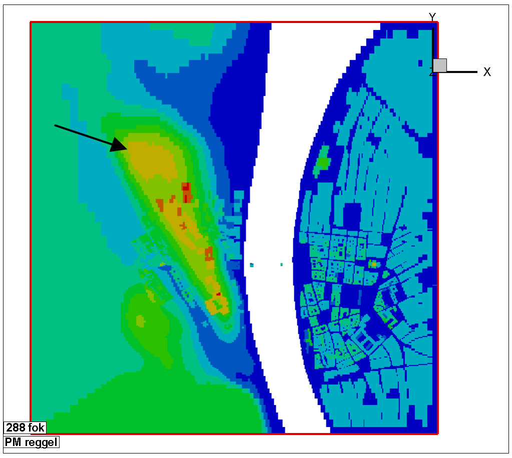
Vortex cores can be extracted based on the theory of critical points (Perry et al. 1990, Haimes et al. 1999). This type of analysis gives directly the vortex filaments in the given frame of reference but there is no information on the extension and strength of vortices. However, the results provide a kind or skeleton of the flow field.

2 The investigated cases

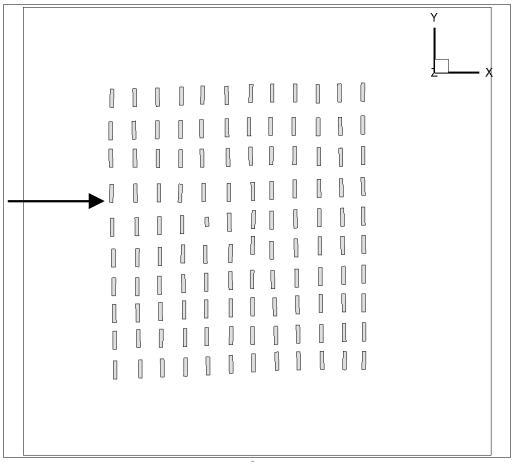
The flow was modeled by Miskam and Fluent past two geometries. The model for Miskam was a part of the downtown of Budapest. This software simplify the real geometry of the buildings significantly, i.e. shapes different from rectangular block cannot be modeled. Roofs and domes are thus missing from the buildings. The models used for the Miskam and Fluent investigations can be seen in Figure 1a and b respectively.

The model where only Fluent was used for flow simulation was a simplified test area consisting of simple blocks arranged in a matrix with slightly randomly varying distances between the blocks (Yee and Biltoft (2004)). The reason why also Fluent simulations were used here, is the possibility of extracting information on the surfaces of the buildings, which is not possible in Miskam, yet. The information on the wall surfaces are the wall-shear vectors that make wall-streakline visualization possible.

In case of the Miskam model the wind direction was 285° in the global orientation system when using the map of Budapest. In this case the hills and mountains in Buda are upstream from the investigated area, thus their effects are included. In the Fluent model the matrix of buildings are blown perpendicular to their longer side faces.



a



b

Figure 1. Investigated models by Miskam (a) and Fluent (b) software

3 Description of numerical modeling

The horizontal extension of the computational domains can be seen in Figure 1. The vertical extension corresponds to 300m height which is the assumed thickness of the atmospheric boundary layer, the models were constructed in the scale of 1:1. The brick shaped domains are characterized by a slip boundary condition on the top surface simulating the free stream off from the boundary layer, a no-slip boundary condition on the wall surfaces, buildings and streets. The sides of the brick domain were of variable boundary conditions to ensure adjusting the direction of wind. Thus the domain had one or two neighboring surfaces of velocity inlet boundary conditions where atmospheric boundary layer flow was prescribed according to the VDI standards. The opposite faces of the computational domain were outlet surfaces where constant static pressure was prescribed.
For all the cases steady state simulations were carried out based on the Reynolds-Averaged Navier-Stokes Equations using turbulence models to take the unsteady effect of turbulence into account. For both software the k-ε turbulence model was used which was found to be appropriate for such large scale flow problems (Franke et al. 2004, Li et al. 2006). Near-wall treatment was solved by means of wall-function approach (with standard logarithmic wall functions).
Throughout all computations constant density was assumed, so no thermal stratification and thus no buoyancy effects were modeled.
Convective and diffusion terms of the governing equations were discretized by second order upwind scheme for Fluent simulations and SIMPLE method was used for pressure-velocity coupling. In case of Miskam computations, however, the discretization was of first order accurate for all terms of the equations.

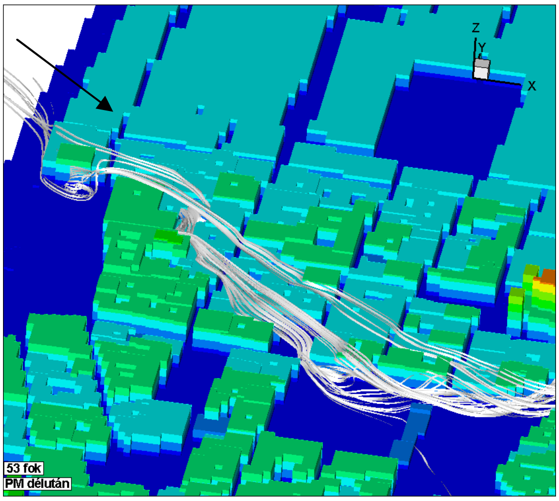
The computational grid consisted of fully hexahedral cells (non-equidistant Cartesian grid in case of MISKAM simulations) for both geometries. In case of Miskam simulations the buildings were simply blocked out from the grid thus it was not body fitted type, while the mesh for Fluent simulations was block structured and body fitted. The number of cells for the Budapest city centre (Miskam) model and for the simplified matrix of buildings (Fluent) were 5million and 1.5million, respectively. Reference velocity was 1m/s (FLUENT) and 10m/s (MISKAM).

4 Interpretation of the flow field

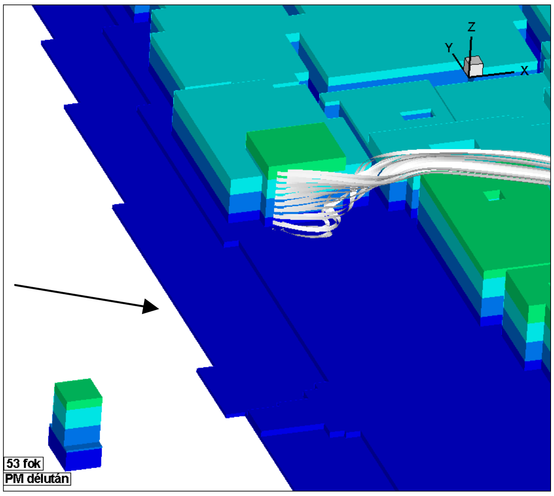
4.1 Flow field represented by streamlines

In the city model only the inner part is detailed and resolved sufficiently thus that region is shown. Some streamlines are shown only to provide ability to follow them through the streets of the city. An overall view and a close-up to the building of the Hungarian Academy of Sciences are represented in Figure 2a and b. As it can be immediately concluded, this way of representation can be effectively used only if small portions of the flow field are visualized. If streamlines were placed over the whole region, no flow structures would be visible. The other conclusion is the well known fact that all the buildings are in interaction with each other and they determine the flow field together, none of them can be handled individually. This is illustrated by the present example where a large vortex with vertical axis is formed beside the entrance of the Hungarian Academy of Sciences (Figure 2b) that involves the sample of the local air quality. The air is transported upwards along the vortex core and is lifted up into the faster upper wind. Streamlines show that this sample of air moves into the courts and gardens of the neighboring group of buildings and more downstream it mixes into the longitudinal “street-canyon”-like vortex along the street on the right lower part of Figure 2a.

Those places where vortices are present might be important from pollution transport aspects as vortices preserve the same air sample for long time along its whole extension. Such a previously mentioned vortex is the street-canyon vortex. In Figure 3 the concentration of pollutants released from the square and the main road passing almost horizontally on the lower half of Figure 2a can be seen. It can be observed that the iso-surface of the concentration of pollutants is surrounded by the streamlines indicating that the pollutants tend to stay inside the vortex extending along the street. On the upstream side of the square in the middle of Figure 3 a ‘backward facing step type’ separation bubble is formed that contains the polluted air. It is known, however, that the vortices forming in the streets are the elements of major influence onto the pollutant propagation in urban environments, differently from the ‘classical’ point-source models over free fields.



a



b

Figure 2. An overall view (a) and the zoom of the region of the Hungarian Academy of Sciences (b) showing streamlines based on Miskam modeling

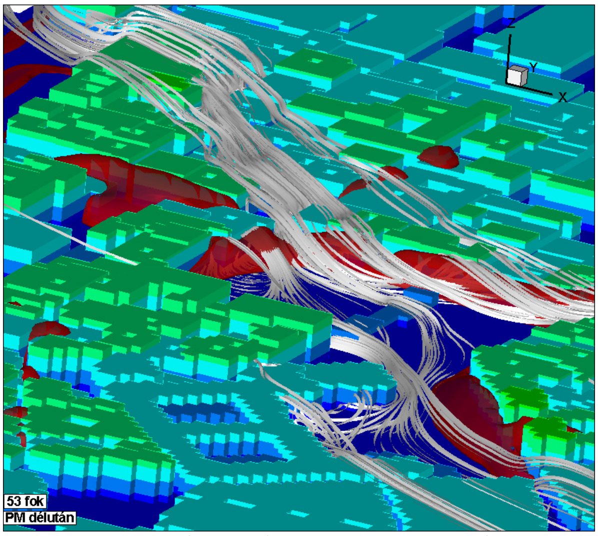
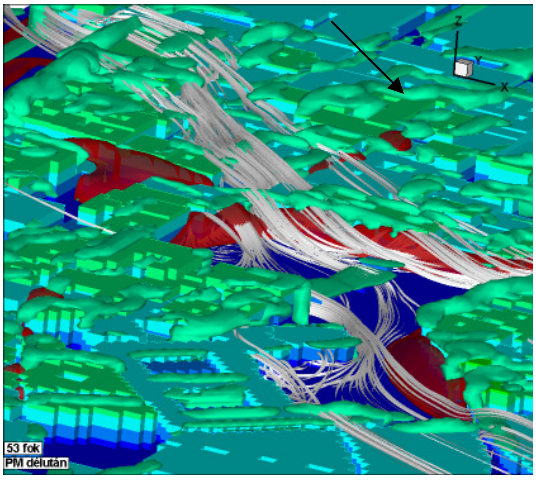


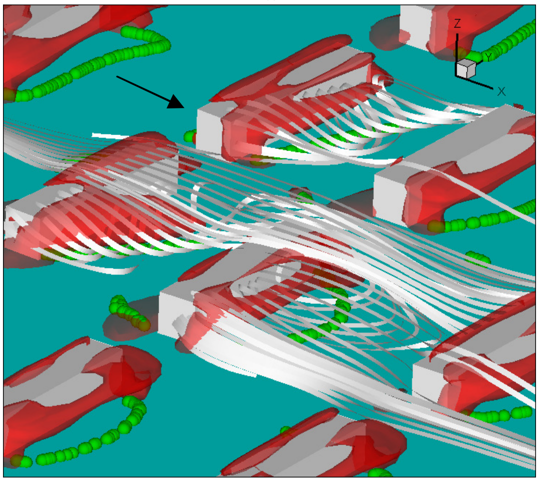
Figure 3 Iso-surfaces of the concentration of pollutants released from the main road and Erzsébet square

4.2 Dynamically significant structures, the iso-Q surfaces

The region of a city is characterized obviously by bluff bodies from fluid mechanics point of view. As the 'local' boundary layer separates from bluff bodies, vortices and recirculation zones are forming. However, in case of a city, an extremely broad size range of vortices are present out of which the largest ones are the street-canyon vortices, the diameter of which is extending the width of the street. These vortices are usually coherent in time and space and can be very well predicted for a certain city and wind direction. The next known size scale is present along the leading upper and side edges of buildings due to the separation of 'local' boundary layer on the individual buildings. The diameters of these vortices are in the order of 1/6th of the building height. These 'small' vortices can be well visualized by the iso-surfaces of the second invariant of the velocity gradient tensor, Q, see Figure 4.



a



b

Figure 4. Iso-Q surfaces over the city model (a) and the simplified matrix of buildings (b). Flow from the upper left corner to the lower right for both figures.

It can be seen in Figure 4 that the iso-Q surfaces that enclose a certain region are located mainly along the upstream upper leading edges of the buildings for both test geometries. The vortices enclosed by iso-Q surfaces are characterized by high rotation rate and are mainly unstable (Jeong and Hussain 1995, Kolar 2006) thus they are usually detaching from the leading edges and shed into the flow. Thus their main role in the mechanism of the flow is 'turbulent mixing', these vortices are the start of the well known turbulent cascade. As these vortices are responsible for the diffusion-like mixing that governs the dispersion of pollutants besides pure convection, the character of their position and overall pattern might provide useful

information on the expectable spreading processes, and as so, an effective tool for influencing it.
Nevertheless, it has to be noted that the iso-Q structures presented here are deduced from a rather coarse computational grid thus their location, size and shape are strictly qualitative. For more accurate prediction of these structures, much finer grids are needed than the present ones.
In Figure 4b on the left hand side an iso-Q surface is surrounded by streamlines that show the formation of a dynamically significant vortex. In this vortex the velocities are expectably high as rotation of fluid particles is higher than their deformation velocity. Similar 'high-Q' vortices produce the lift for delta-wing airplanes.
Also vortex cores, represented by spheres can be seen in Figure 4b. These vortex cores are shifted from the centers of vortices mainly due to the coarse grid but practically should be placed exactly in the axis of them.

4.3 Wakes, iso-surfaces of total pressure

The separation bubbles that are well captured by streamline visualization are mainly characterized by very low velocity comparing to the free stream values. In these 'dead regions' vorticity is almost negligible thus the Q property is not applicable to detect them. Mainly due to losses, total pressure decrease inside them, thus iso-surfaces of it can enclose recirculation regions. Total pressure was obtained for the simplified matrix of buildings and is represented in Figure 5.

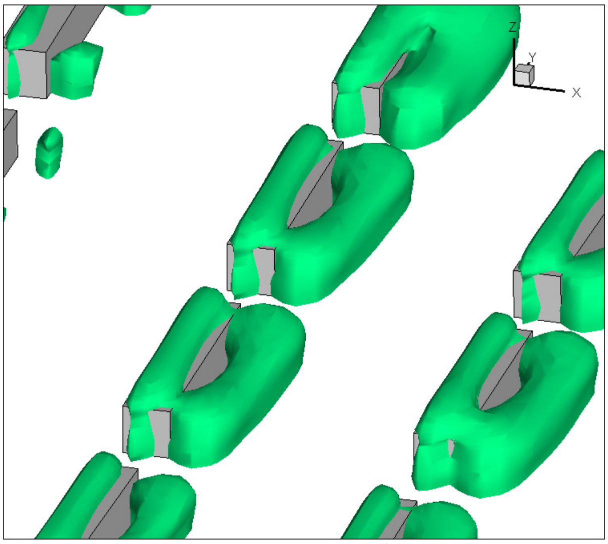


Figure 5. Iso-total pressure surfaces around the buildings for the simplified matrix arrangement

The iso-surfaces of total pressure enclose the whole region behind the building models where recirculation was present, as well, as regions which were also detected by the Q property.

4.4 Wall streaklines

Traces of vortices and main flow directions can be determined from the 'surface flow pattern' which is based on the shear forces exerted on the surfaces by the flow. Such a surface flow pattern can be seen in Figure 6.
In Figure 6 also vortex cores are represented that help in understanding the flow pattern. The surface flow is in relation with the local discomfort parameters as this region of the flow field is in direct connection with the pedestrians. From, for example, sand-erosion experiments one can map the regions characterized by high wall shear but the flow structures producing these high values are remain 'invisible'. By applying the methods shown above simultaneously, more detailed information on the flow field can be gathered and the main flow mechanisms can be understood. Understanding the flow field might help in the determination of influencing it effectively.

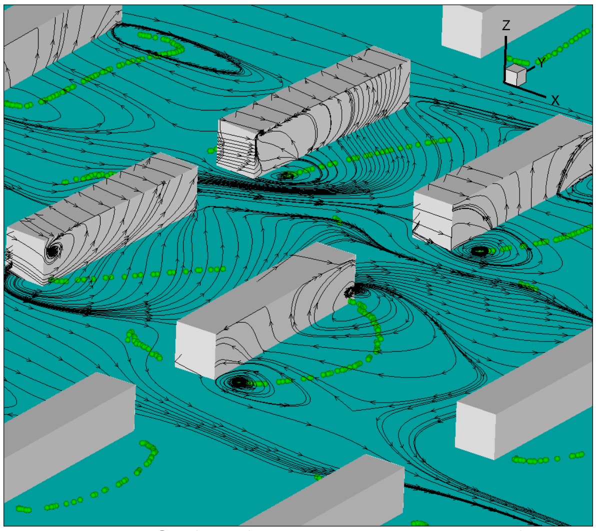


Figure 6. Surface flow pattern in a portion of the simplified building matrix

5 Conclusion

In this paper different flow visualization methods were shown that form a useful tool for understanding the flow field around complicated geometries, like urban environments. For the investigation of the flow field two city models were applied and the flow was determined via numerical modeling. The model that represented the inner city of Budapest was used for numerical simulations by software Miskam. On this model the wall surfaces did not contain any information thus characteristics regarding to the volume of the flow field were extracted only. To show additional information originating from the wall surfaces flow patterns an additional simplified city model was applied over which the flow was simulated by software ANSYS-Fluent. Surface flow patterns are in strong relation with the discomfort parameters of the pedestrians.
The flow field was visualized by different properties deduced from the variables.

Acknowledgments

Authors of this paper wish to express their thanks for NKFP 3A/088/2004 for providing the financial basis to make these investigations possible.

References

Adrian, R. J. (2007) "Hairpin vortex organization in wall turbulence", Physics of Fluids, Vol. 19, Issue 4, pp. 041301-041301-16

Dezső-Weidinger, G., Stitou, A.; van Beeck, J., Riethmuller, M. L. (2003) "Measurement of the turbulent mass flux with PTV in a street canyon" Journal of Wind Engineering and Industrial Aerodynamics, vol. 91, pp. 1117-1131

Franke, J., Hirsch, C., Jensen, A. G., Krüs, H. W., Schatzmann, M., Westbury, P.S., Miles, S.D., Wisse, J.A., Wright, N. G. (2004)

"Recommendations on the use of CFD in wind engineering", Proceedings of the International Conference on Urban Wind Engineering and Building Aerodynamics, VKI, Belgium, May 5-7, pp. C.1.1-C.1.11

Gromke, C., Ruck, B. (2007) "Influence of trees on the dispersion of pollutants in an urban street canyon--Experimental investigation of the flow and concentration field" Atmospheric Environment, 2007 , 41 , 3287-3302

Haimes, R., Kenwright, D. (1999) "On the velocity gradient tensor and fluid feature extraction", AIAA paper No. 99-3288

Hussain, A.K.M.F. (1986) "Coherent structures and turbulence", Journal of Fluid Mechanics. Vol. 173, pp. 303-356

Jeong, J., Hussain, F. (1995) "On the identification of a vortex", Journal of Fluid Mechanics, vol. 285, p. 69-94

Kolar, V. (2006) "On universal vortical features of jets in crossflow", Proceedings of Physmod 2006 conference, pp. 8-9

Li, X., Liu, C., Leung, D. Y., Lam, K. (2006) "Recent progress in CFD modelling of wind field and pollutant transport in street canyons", Atmospheric Environment, vol. 40, pp. 5640-5658

Lohász, M. M. Rambaud, P. Benocci, C. (2006) "Flow Features in a fully developed ribbed Duct Flow as a Result of MILES", Flow, Turbulence and Combustion, 77:59-76

Mathur, M., Haller, G., Peacock, T., Ruppert-Felsot, J. E., Swinney, H. L. (2007) "Uncovering the Lagrangian skeleton of turbulence", Phys. Rev. Lett. 98, 144502

Perry, A.E., Chong, M.E., Cantwell, B.J. (1990) "A general classification of three-dimensional flow fields", Physics of Fluids A 2 (5), May, 765-777

Schafer, K.; Emeis, S.; Hoffmann, H.; Jahn, C., Muller, W. J., Heits, B., Haase, D., Drunkenmolle, W. D., Bachlin, W., Schlunzen, K. H., Leitl, B., Paschek, F. Schatzmann, M. (2005) "Field measurements within a quarter of a city including a street canyon to produce a validation data set", International Journal Of Environment And Pollution, vol. 25, pp. 201-216

Uehara, K., Murakami, S., Oikawa, S. Wakamatsu, S.(2000) "Wind tunnel experiments on how thermal stratification affects flow in and above urban street canyons" Atmospheric Environment, vol. 34, pp. 1553-1562

Plate, E.J. (1982) "Engineering Meteorology", ed. E.J. Plate, Elsevier Scientific Publishing Company, Amsterdam-Oxford-New York

VDI Guideline 3783 Part 12 (2000) "Environmental meteorology, Physical modelling of flow and dispersion processes in the atmospheric boundary layer, Application of wind tunnels", VDI/DIN-Handbuch Reinhaltung der Luft, Band 1b

Yee, E., Biltoft, C. A. (2004) "Concentration fluctuation measurements in a plume dispersing through a regular array of obstacles", Boundary-Layer Meteorology, Kluwer Academic Publ, vol. 111, pp. 363-415