Two Cases of the Real ABL Flow over Complex Orography

L. Beneš, T. Bodnár, K. Kozel, I. Sládek
Department of Mathematics, Czech Technical University, Karlovo náměstí 13, CZ-121 35 Praha 2, Czech Republic

The air pollution resulting from a rapid industrialization has become a serious environmental problem mainly in the North Bohemia region. Two concrete cases of flow and pollution dispersion over a complex topography are presented. In the first case, the influence of several types of obstacles (e.g. solid wall, forest block and shelter belt) on the dustiness of the coal depot is numerically simulated.In the second case, the impact of the selected pollution sources to the east part of the Giant mountains is studied.

Mathematical Model

The flow in ABL is assumed to be viscous, steady, incompressible, turbulent and indifferently stratified. The mathematical model is based on Reynolds Averaged Navier–Stokes equations. The artificial compressibility method is used for the numerical solution.

$$\begin{pmatrix} \frac{p}{\beta^2} \\ u \\ v \\ w \\ C \end{pmatrix}_t + \begin{pmatrix} u \\ u^2+p \\ uv \\ uw \\ uC \end{pmatrix}_x + \begin{pmatrix} v \\ uv \\ v^2+p \\ vw \\ vC \end{pmatrix}_y + \begin{pmatrix} w \\ wu \\ wv \\ w^2+p \\ wC \end{pmatrix}_z = \begin{pmatrix} 0 \\ Ku_x \\ Kv_x \\ Kw_x \\ C_x \end{pmatrix}_x + \begin{pmatrix} 0 \\ Ku_y \\ Kv_y \\ Kw_y \\ C_y \end{pmatrix}_y + \begin{pmatrix} 0 \\ Ku_z \\ Kv_z \\ Kw_z \\ C_z \end{pmatrix}_z + f_v$$

where p is the pressure, three velocity components $V = (u, v, w)^T$, and C denotes the concentration of passive pollutant. Further f_v denotes the volume force, β artificial compressibility coefficient and finally K represents the turbulent diffusion coefficient. A simple algebraic turbulence model designed for ABL flow was used

$$K = \nu + \nu_T \qquad \nu_T = l^2\sqrt{(u_z)^2 + (v_z)^2} \qquad l = \frac{\kappa(z+z_0)}{1+\kappa\frac{(z+z_0)}{l_\infty}} \tag{1}$$

where ν_T, ν are turbulent and laminar viscosities, κ is the von Karman constant, λ denotes the Coriolis parameter, z_0 the roughness length, l_∞ denotes the mixing length for $z \to \infty$.
For comparison, some results were also calculated using standard high Re $k - \varepsilon$ turbulence model.

Numerical Methods

Two different robust numerical methods based on FV and FD have been used. A structured non-orthogonal grids made of hexahedral (3D–case) and quadrilateral (2D–case) control cells are applied.

Finite volume method: The finite volume method (cell-centered type) of the following form was used

$$W_t\Big|_{ijk} \approx -\frac{1}{\mu_{ijk}}\sum_{1}^{6}(F - KR, G - KS, H - KT)|_{i\pm\frac{1}{2},j\pm\frac{1}{2},k\pm\frac{1}{2}} dS \tag{2}$$

The value on the cell face is calculated by central interpolation. For instance

$$F_{i+\frac{1}{2},j,k} = ((u)_{i+\frac{1}{2}}, (u^2+p)_{i+\frac{1}{2}}, (uv)_{i+\frac{1}{2}}, (uw)_{i+\frac{1}{2}}, (uC)_{i+\frac{1}{2}})$$

and quantity q is calculated as $q_{i+\frac{1}{2}} = (q_i + q_{i+1})/2$. The viscous fluxes are discretized in the same way. Derivatives on cell faces are calculated using similar treatment on the dual mesh (diamond type). For the integration in the dual time a (3)–stage explicit Runge–Kutta scheme has been applied.
The finite volume scheme has been validated through the ERCOFTAC's test–case of fully developed channel flow over 2D polynomial–shaped hill mounted on a flat plate. The Almeida's experimental and the ERCOFTAC's $k - \varepsilon$ reference numerical data have been used for the comparison.

Finite difference method: The second numerical method is based on a semi-implicit finite difference scheme. The left-hand side of momentum equations is discretized in this way

$$\mathsf{V}_t \sim \overrightarrow{\delta_t}\mathsf{V}^n_{i,j,k} \qquad u\mathsf{V}_x \sim \frac{1}{2}\left(u^n_{i+1/2}\overrightarrow{\delta_x}\mathsf{V}^n_{i,j,k} + u^n_{i-1/2}\overleftarrow{\delta_x}\mathsf{V}^{n+1}_{i,j,k}\right) \tag{3}$$

$$v\mathsf{V}_y \sim \frac{1}{4}\left\{\left(v^n_{j+1/2}\overrightarrow{\delta_y}\mathsf{V}_{i,j,k} + v^n_{j-1/2}\overleftarrow{\delta_y}\mathsf{V}_{i,j,k}\right)^n + \left(v^n_{j+1/2}\overrightarrow{\delta_y}\mathsf{V}_{i,j,k} + v^n_{j-1/2}\overleftarrow{\delta_y}\mathsf{V}_{i,j,k}\right)^{n+1}\right\}$$

$$w\mathsf{V}_z \sim \frac{1}{4}\left\{\left(w^n_{k+1/2}\overrightarrow{\delta_z}\mathsf{V}_{i,j,k} + w^n_{k-1/2}\overleftarrow{\delta_z}\mathsf{V}_{i,j,k}\right)^n + \left(w^n_{k+1/2}\overrightarrow{\delta_z}\mathsf{V}_{i,j,k} + w^n_{k-1/2}\overleftarrow{\delta_z}\mathsf{V}_{i,j,k}\right)^{n+1}\right\}$$

Here $\overrightarrow{\delta}$, $\overleftarrow{\delta}$ denotes backward and forward differences. The arising 5–diagonal system of equations is transformed into the 3–diagonal and solved iteratively in the plane $i = const$. Then, the pressure is updated from the modified continuity equation $p_t + \beta^2(u_x + v_y + w_z) = 0$. For its discretization, similar treatment is used.
The method has been validated on the experimental and reference numerical data obtained by G.H.Kim. The flow over the hills of different shapes and over the real topography of Korea has been compared.
Both methods are second order accurate both in space and time and they are stabilized by the artificial viscosity term of the Jameson type. In order to minimize its negative influence, only fourth order terms were used. Optimal convergence we have obtained for $\beta \in < 2, 4 >$ (in normalized case).

Obstacle modeling

We have tested several types obstacles.
• **Forest block:** for the forest block, the force vector $\vec{f_v}$ includes the specific aerodynamic force corresponding to the drag induced by the vegetation, i.e.

$$\vec{f_v} = col(-r_h|V|u, -r_h|V|v, -r_h|V|w)^T \quad r_h(z) = \begin{cases} r\,\frac{z/h}{0.75} & \text{for } \; 0 \le z/h \le 0.75 \\ r\,\frac{1-z/h}{1-0.75} & \text{for } \; 0.75 \le z/h \le 1.0 \end{cases} \tag{4}$$

where the drag coefficient value r is given a priori.
• **Solid wall:** this type of obstacle is simulated by a column of cells in which the velocity components is set simply to zero.
• **Shelter–belt:** represents a sudden solid obstacle in the flow field. It has a rectangular form with angles almost 90 degrees at some corners. Two different strategies was used for its simulation.

1. Classical body–fitting mesh. There are regions where the mesh cells are strongly distorted. Another problem is, that the shelter belt of different dimensions and angles require construction of different grids.
2. Zero Velocity Obstacle Model (ZVOM). A rectangular computational domain is non-uniformly griddet. Cells located within the obstacle are identified and all the three velocity components are set to zero. In order to respect the Prandtl's assumption of a constant pressure over the boundary layer, the pressure from the neighbor cells is copied to the internal cells lying on the obstacle border.

Boundary conditions

• Inlet: Giant mountains $u = \frac{u_*}{\kappa}ln(\frac{z}{z_0})$, friction velocity $u_* = 0.4$, coal depot $u = U_0(z/L)^\alpha$, $v = w = C = 0$, where L is height of the domain. Both $v = w = C = 0$.
• Outlet: $u_x = v_x = w_x = C_x = 0$.
• Wall: the no–slip condition for the velocity components, $\frac{\partial C}{\partial n} = 0$.
• Top: $u = U_0$, $v = 0$, $\frac{\partial w}{\partial z} = \frac{\partial C}{\partial z} = 0$.
• Sides: homogenous Neumann conditions.
• Pressure on the boundary is linearly extrapolated from the flow field and a value in a one point is given.

Numerical Results

Brown coal depot:

The major task was to design a safety obstacle close to the coal depot. Several types of obstacles such a solid wall, protective tree line, forest block and shelter belt were modeled. We have used two imbedded domains. The data obtained on the Domain 1 was used as the boundary and initial conditions for the computations on Domain 2.
Domain 1: 800×480 m,the upper side is at 1000 m. $100 \times 60 \times 40$ mesh cells. The coal depot has dimensions 80×20 m and it is situated at the origin.
Domain 2: 400×240 m, $100 \times 60 \times 40$ mesh cells, the horizontal resolution is $4m$, $\Delta_{z_{min}} \approx 0.6m$.
The third configuration is a 2D cut. From Domain 1 the 2D middle section-cut ($y = 0$) is chosen and discretized by 800×40 cells (horizontal resolution $1m$), vertical distribution is the same as in 3D. Mean free stream velocity $U = 10\,m/s$.

Computed cases (both methods were compared):
• In the 3D eight variants were calculated. Basic situation without protective obstacles, with two forest blocks situated one before and another behind the coal depot, and six situations with rampart at an angle 90° and 45° to flow directions and with combinations of two ramparts. Obstacle height varies from 3 to 12m.
• Ten different positions and combinations of walls, ramparts and forests were calculated in the 2D case: basic–without obstacles (zak), with the forest before, behind and on both sides of the depot, with the wall before, behind and on both sides and with rampart behind coal depot. The forest blocks are 10m high with the drag coefficient $r = 0.19$. The wall is 5m high. The rampart is 3, 5, 7m high.

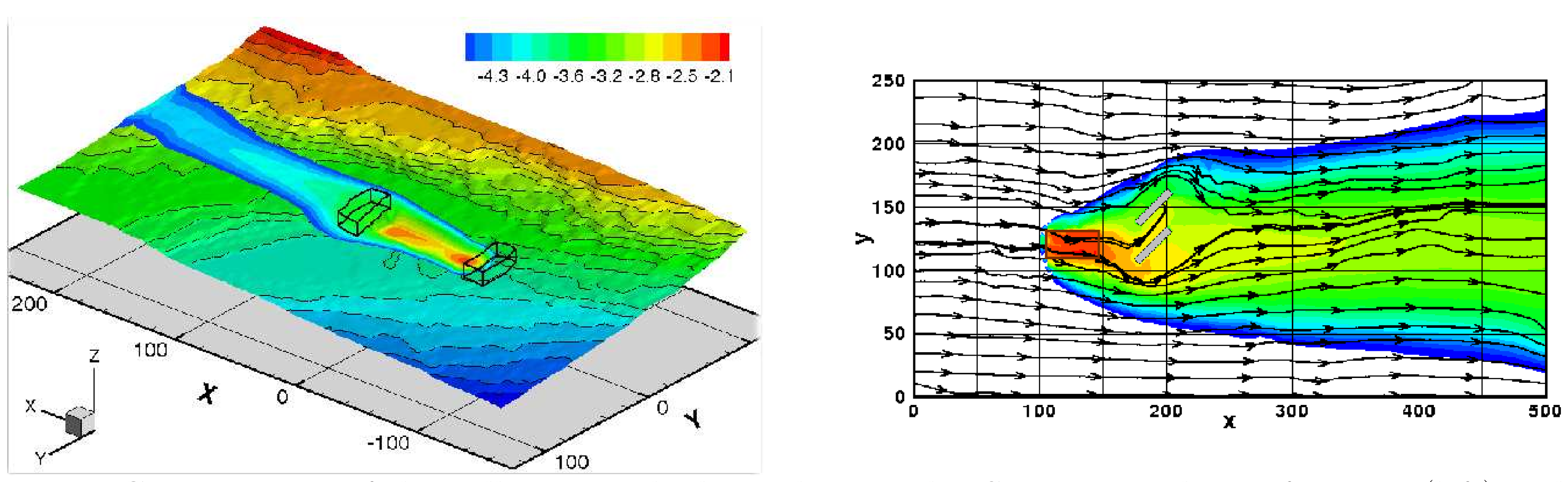

Figure 1: Concentration of the pollution in the logarithmic scale. Situation with two forests – (left) and with double rampart (right).

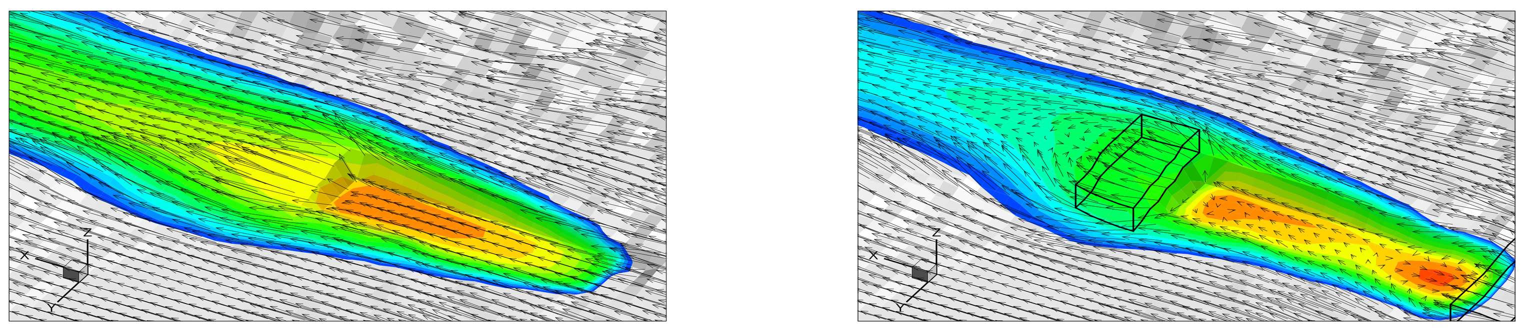
Figure 2: Velocity vectors and concentration close to the coal depot- basic variant (left) and situation with two forests (right).

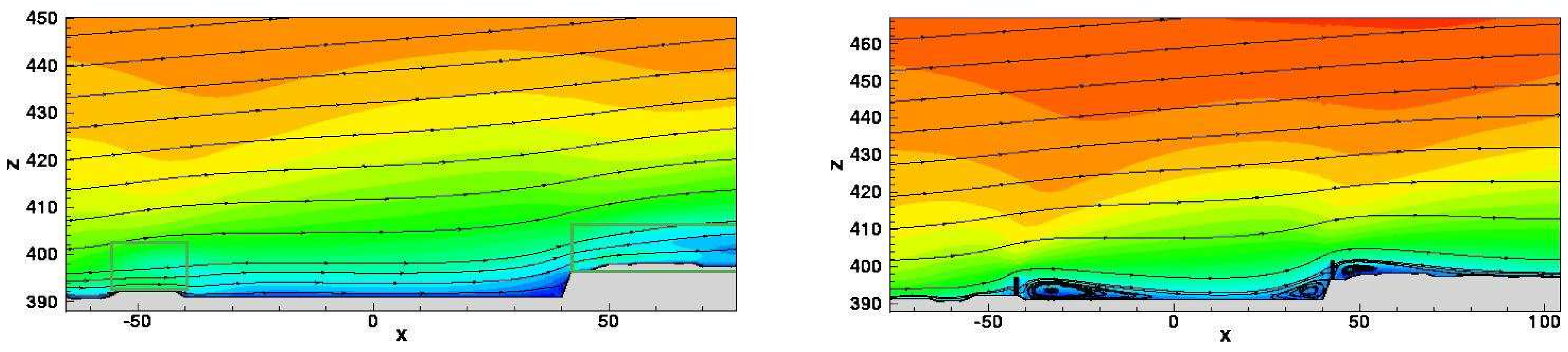

Figure 3: Variant ab (left) and fab (right) - u velocity component with streamlines close to the c. d.

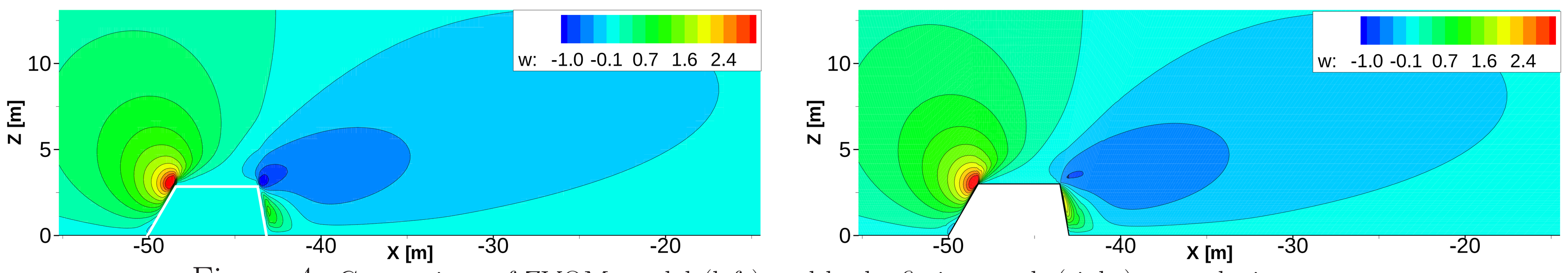

Figure 4: Comparison of ZVOM model (left) and body–fitting mesh (right) - w velocity component.

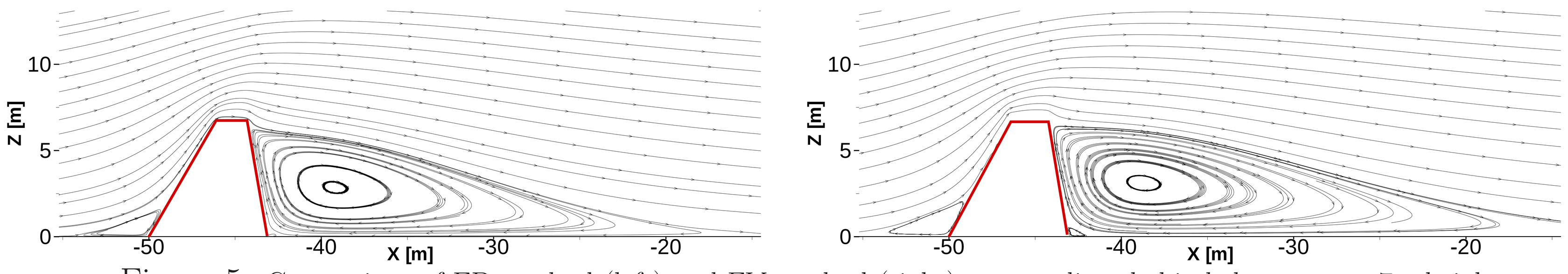

Figure 5: Comparison of FD method (left) and FV method (right) - streamlines behind the rampart 7m height.

Giant Mountains:

Methods have been used to calculate the velocity flow field and first of all the concentration map of the passive pollutants emanating from four point sources over the topography of the Giant mountains.
Computational domain Ω is 30x24x2km, discretized either by 60x48x30 or 120x96x30 cells in order to study dependence on mesh. The mesh is exponentially refined close to the ground with minimal $\Delta z = 0.5m$. Four real stationary point sources are assumed. Two different wind flow directions are supposed: direction inclined about a) 90° and b) 120° from the x axis. The other parameters are: the mean free stream velocity $U \sim 10\,\text{m/s}$, Reynolds number $Re = U \cdot L/\nu = 1.3 \cdot 10^8$, the roughness parameter $z_0 = 0.2\,\text{m}$.

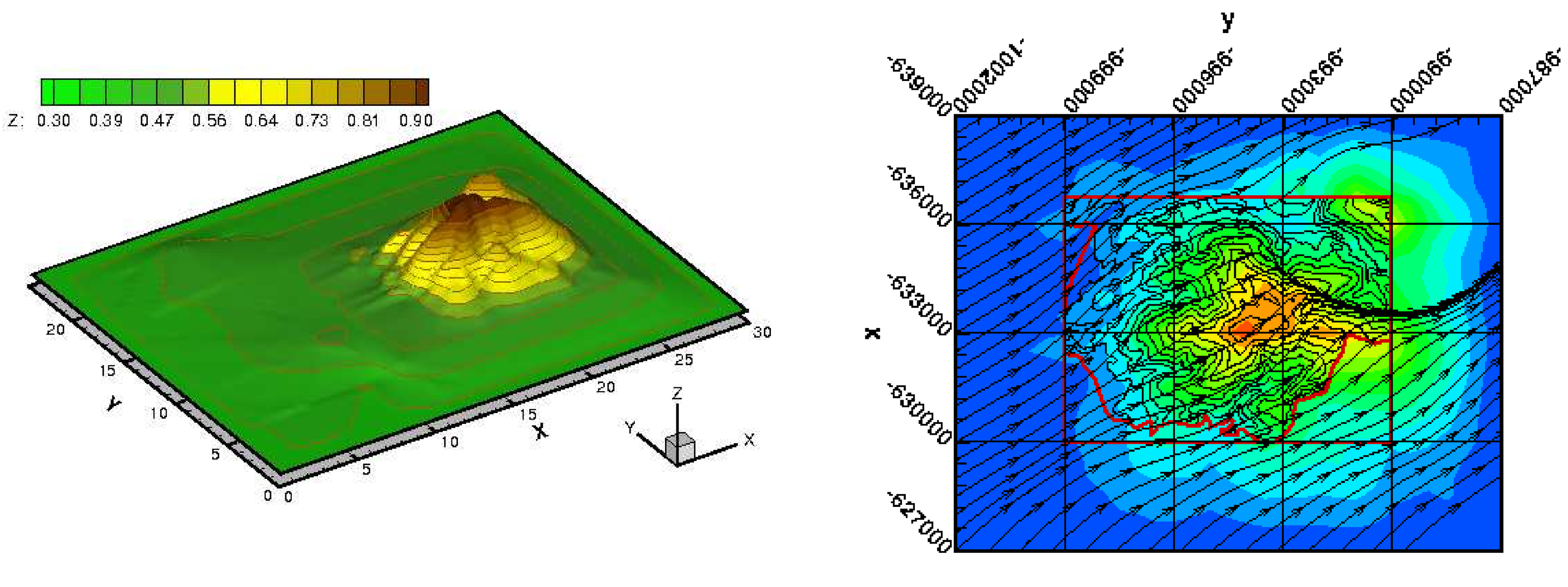

Figure 6: Topography of the mountains (left). Near ground streamlines (right), wind direction 120°.

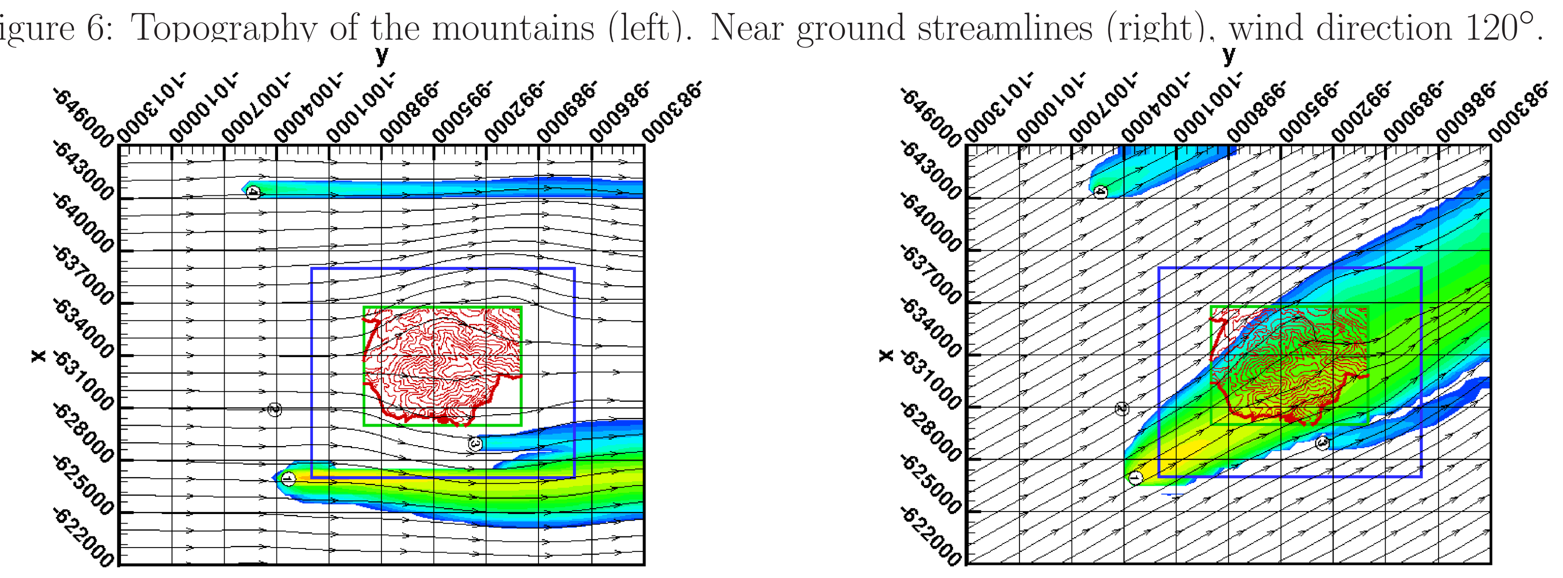

Figure 7: Concentration (in log-scale) of passive pollutants with superimposed near ground streamlines. Wind direction 90° (left) and 120° (right).

Acknowledgment: The financial support for the project is partly provided by the Research Plan MSM 6840770003 and grants GAAS 1ET400760405 and GAČR 205/06/0727.

References

[1] T. Bodnár, Ph. Fraunié, K. Kozel, I. Sládek: Numerical simulation of Complex Atmospheric Boundary Layer Problems,*ERCOFTAC Bulletin No.60, March 2004, p.5-12*

[2] T. Bodnár, I. Sládek, E. Gulíková: Numerical Simulation of Wind Flow in the Vicinity of Forest Block *Advances in Computational & Experimental Engineering & Sciences. Forsyth: Tech Science Press, 2004, s. 554-559. ISBN 0-9657001-6-X.*

[3] L. Beneš, T. Bodnár, I. Sládek, K. Kozel: Numerical study of atmospheric flow over open coal mine. *9th ICFD Conference on Numerical Methods for Fluid Dynamics, Reading 2007*