Transactions of the VŠB – Technical University of Ostrava, Mechanical Series
No. 2, 2010, vol. LVI
article No. 1777

Tomáš BLEJCHAŘ[*], Vladimíra MICHALCOVÁ[**]

CFD SIMULATION IN BOUNDARY LAYER IN COAL STOCKPILE SURROUND

SIMULACE PROUDĚNÍ V MEZNÍ VRSTVĚ ATMOSFÉRY V OKOLÍ UHELNÉ SKLÁDKY

Abstract

The paper deals with formulation of CFD simulation, which describes fluid flow in atmospheric boundary layer in coal stockpile surround. The paper presents preliminary phases of global project, which aim to spontaneous ignition of coal matter in coal stockpile and coal dump. Simulations of flow in boundary layer include only dominant direction of wind. First part of the global task is focused on generation of Earth's surface topology on the basis of GIS data. Second task is focused on simplification of simulation domain and reduction of mesh size by substitution of mine building by porous subdomain. At the end of this paper are presented results of CFD simulation in 4.3 x 4.3 x 0,4 km domain with coal stack and mine buildings.

Abstrakt

Článek popisuje problematiku tvorby CFD modelu, který simuluje proudění vzduchu v mezní vrstvě atmosféry v okolí uhelné skládky. Tato úloha představuje první fázi globálního problému, který je zaměřen na problematiku rozvoje oxidace a samovzněcování uhelné hmoty obsažené v uhelných skládkách a odvalech umístěných na povrchu. Jedná se řešení globálního proudění v okolí haldy pro převažující směry větru. První část se zabývá problematikou tvorby topologie terénu z geografických dat. V druhé části úlohy je řešen také proces zjednodušování oblasti, které spočívá v nahrazení budov oblastí s odporovými koeficienty. V závěru jsou prezentovány výsledky modelování proudění v oblasti 4.3 x 4.3 x 0.4 km, která zahrnuje, uhelnou skládku a budovy dolu.

1 INTRODUCTION

Flow in boundary layer is characteristic by time depending of velocity magnitude and direction vector of wind. The spontaneous ignition of coal matter in coal stockpile and dump is strongly dependent on wind velocity. Flow in coal stockpile or dump is laminar and flow in boundary layer is fully turbulent. Then the simulation includes two diametrically different regimes of flow. Simulation in atmospheric boundary layer is very difficult and complexity, so a lot of simplifications must be accepted for reasonably CFD simulation.

2 TOPOLOGY OF EARTH'S SURFACE

CFD simulation perform in 4,3 x 4,3km and high 400m domain. This domain includes real Earth's surface, mine buildings and coal stack, Fig.1. The mine and coal stack is situated in Orlová Lazy near city Karviná. Domain included entering surfaces, which were used to unification of air inlet and outlet altitude and flow area respectively. This modification make possible the definition of turbulent quantities. The mesh size ratio and cell size near the wall must be observed, because turbulent quantities equilibrium is strongly dependent on quality of mesh [1], see Fig.2 right. The mesh size is relatively extensive (ca 1,5mil. cells) if the mesh criteria are within accepted limits. The num-

[*] Ing., Ph.D., Department of Hydrodynamics and Hydraulic Equipment, Faculty of Mechanical Engineering, VŠB – Technical University of Ostrava, 17. listopadu 15, Ostrava - Poruba, 708 33, Czech republic tel. (+420) 59 732 5753, e-mail tomas.blejchar@vsb.cz

[**] Ing.,Ph.D.,, Department of Structural Mechanics, Faculty of Civil Engineering, VŠB – Technical University of Ostrava, L. Poděště 1875/17, Ostrava - Poruba, 708 33, Czech republic tel. (+420) 59 732 1348, e-mail vladimira.michalcova@vsb.cz

ber of cells must be smaller with respect to future implementation of coal oxidation and spontaneous ignition.

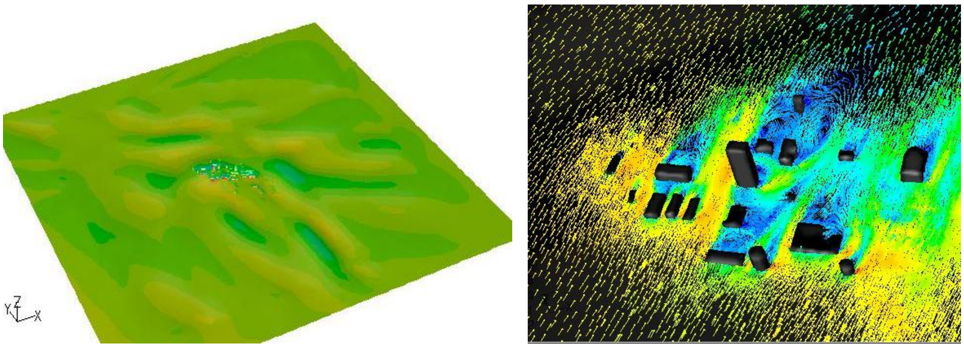

Fig. 1 Computation domain with real Earth's surface near mine Lazy, domain includes mine buildings too (left) and velocity vector near mine buildings (right)

Mine buildings can be substituted by simply porous domain, which has pressure drop equal to origin domain with buildings see Fig.2 left. Complicated geometry of the buildings was replaced by simply porosity volume. Resistant coefficient of the porous domain was specified by means of thin domain near the buildings. Pressure resistance coefficient is calculated by means of pressure drop and change of reference velocity of wind for real geometry with buildings. Results of the calculation is resumed in Tab.1.

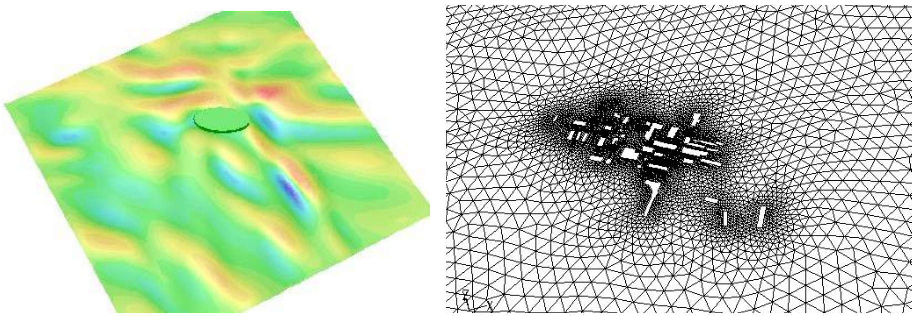

Fig. 2 Simplified computation domain with porous subdomain which substituted mine buildings (left) and origin grid near mine buildings (right)

Tab.1 Resistance coefficient calculation

Inlet velocity	Outlet velocity	Velocity near buildings	Static pressure on boundary		Pressure difference Δp	Pressure drop	Lost coeff..	Resistance coeff.
$\left[\frac{m}{s}\right]$			[Pa]				[-]	$\left[\frac{1}{m}\right]$
17.4	22.7	21.3	378.2	72.8	305.4	176.17	0.632	0.0021
11.6	15.1	14.2	167.2	32.3	134.9	77.43	0.625	0.0021
5.8	7.6	7.1	42.2	7.8	34.4	19.98	0.645	0.0022
2.3	3.0	2.8	6.9	1.4	5.6	3.26	0.677	0.0023

The computation mesh have to include inflate boundary layer for detail simulation of wind flow from Earth's surface up 10 m, because coal stockpile high is approximately 10 m. Inflate boundary layer cannot be created via standard tools implemented in mesh creator. Therefore 10 m equidistant surface is in computation domain. This surface split the domain into new two separately domain. In new thin 10 m high domain is created mesh per cooper schema with grow ratio 1.25, see Fig.3 and 4.

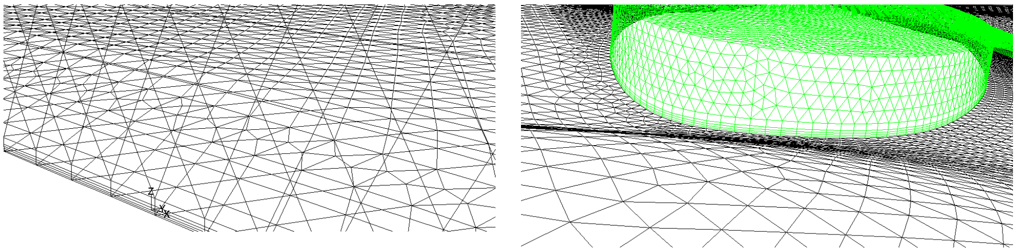

Fig. 3 Mesh with inflate boundary layer

Density mesh is high around the domains which represent buildings and coal stockpile too. This mesh is relatively high, mesh size is ca 1.7 million. Next problem is crack located into coal stockpile. The crack is in order centimetres. This crack is not included into simulation on the present. Including of the crack into geometry will cause dramaticaly increase of the mesh size. This is reason why the new mesh size decreasing method is proposed.

The new method is divided into two basic steps

1) Simulation of flow in atmospheric boundary layer will specified for full domain 4.3 x 4.3 x 0.4 km, and coal stockpile will not include the crack.
2) Increase of the mesh size
3) Domain will be split by boundary surfaces (see Fig. 4). New subdomain size will be 500 x500 x 400 m with coal stockpile in centre of the subdomain.
4) Into the new subdomain will be added the crack in coal stockpile. This simulation will be included chemical reaction, which describes low temperature oxidation and spontaneous ignition of coal matter.
5) Transfer of boundary condition will be preformed via boundary profiles. The profiles will be written on boundary surfaces in full simulation (step1). Then the profiles will be read in small simulation with subdomain

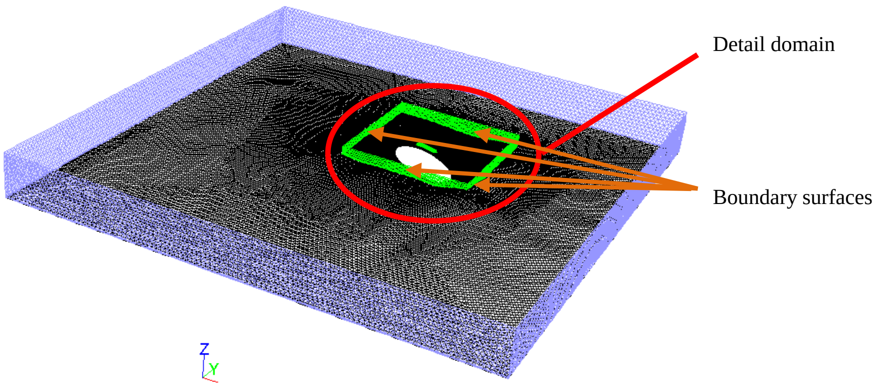

Fig. 4 Final mesh with new boundary surfaces

3 MATHEMATIC MODEL OF FLOW IN ATHMOSPHERY BOUDARY LAYER

The basic equation set which describes the bought regimes of flow represents application of conservation law. The law of mass, momentum and energy conservation is used to numerical modelling of flow. The law of momentum conservation represents Navier-Stokes equation. The Law of mass conservation represents continuity equation, and the law of energy equation represent energy equation. For unsteady compressible and non isothermal flow are the equations defined by following equations [2], [3]. Turbulent flow is stochastic, but it is statistic stable. Arbitrary physical quantity can be decompiled on sum of average value and fluctuation value $u = \overline{u} + u'$. Commonly used turbulent models are based on description of local turbulent intensity by length and velocity scale. Standard k – ε turbulent model was used to CFD simulation in this case. The turbulent viscosity is specified by two transport equations for turbulent kinetic energy k and dissipation e. [2],[3]

4 BOUNDARY CONDITIONS

CFD simulation is calculated with mesh see Fig 4. Only three dominant directions of the wind are simulated. Three dominant directions of the wind are South, South-East, South-West. Velocity profile in boundary layer is described by power function, and temperature profile is described by linear function. Atmosphere is stable, so the temperature of air degreasing with altitude. The air compressibility influence is needed for boundary condition definition, because the domain high is inconsiderable and density of air is function of altitude.

Velocity profile is defined by power function (1) with reference velocity $v_{ref} = 5$ m/s. Reference velocity is defined in $z_{ref} = 10$ m altitude. Power coefficient for stable state of atmosphere is $p = 0.14$

$$v = v_{ref} \cdot \left(\frac{z}{10} \right)^p \quad (1)$$

where:

z – altitude $[\mathrm{m}]$,

v_{ref} – reference velocity in 10 m altitude $\left[\frac{\mathrm{m}}{\mathrm{s}} \right]$,

p – power coefficient $[-]$,

Temperature profile is defined by linear function (2), which describes degreasing of temperature as function of altitude. Temperature of Earth's surface is chosen $t_0 = 13$ °C and $T_0 = 286$ K respectively . Temperature gradient for stable state of atmosphere is $g = -0.0065$ K/m.

$$T = T_0 - g \cdot z \quad (2)$$

where:

T_0 – temperature of Earth's surface $[\mathrm{m}]$,

γ – temperature gradient $\left[\frac{\mathrm{K}}{\mathrm{m}} \right]$,

Atmospheric pressure is defined by function which includes influence of air temperature and acceleration of gravity. So the simulations include aerostatic pressure. Preference pressure $p_0 = 101325$ Pa is defined as standard atmospheric pressure on the sea level.

$$p = p_0 \left(1 - \frac{g \cdot z}{T_0}\right)^{\frac{-g}{r \cdot g}} \tag{3}$$

where:

p_0 – standard atmospheric pressure $[\mathrm{Pa}]$,

g – acceleration of gravity $\left[\frac{\mathrm{m}}{\mathrm{s}^2}\right]$,

r – gas constant of air $\left[\frac{\mathrm{J}}{\mathrm{kg} \cdot \mathrm{K}}\right]$,

Turbulent kinetic energy k and dissipation e are defined by frictional velocity $v_* = 0.2258$ and altitude

$$e = \frac{{v_*}^3}{0.4 \cdot z} \tag{4}$$

$$k = \frac{{v_*}^2}{0.3} \tag{5}$$

where:

v_* – frictional velocity $\left[\frac{\mathrm{m}}{\mathrm{s}}\right]$,

Previous formulas is used for formulation of boundary profiles on inlet and outlet. Inlet of domain is defined by means of mass flow and direction vector. Outlet of domain is defined by means of static pressure. Surfaces parallel to velocity direction are defined as symmetry boundary condition see Fig.5 and Fig. 6.

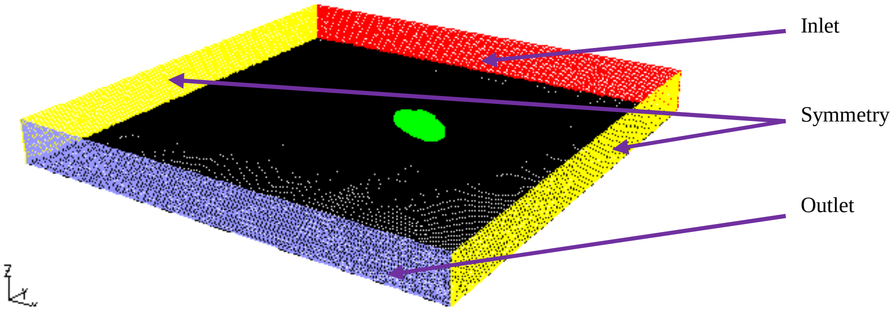

Fig. 5 Boundary conditions set for South direction of wind

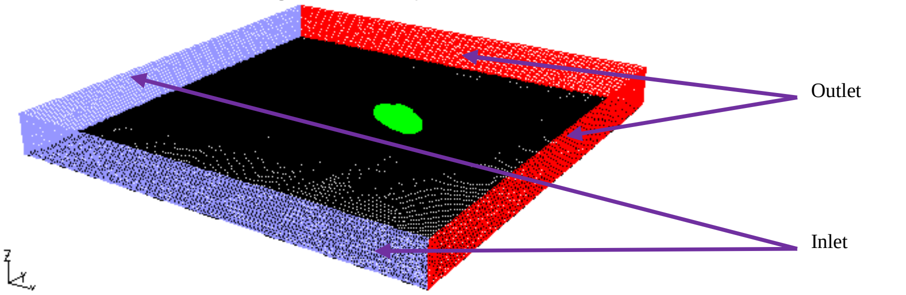

Fig. 6 Boundary conditions set for South-West direction of wind

5 RESULTS

Main results of first simulations are aerodynamic tension on coal stockpile surface, see Fig. 7. Aerodynamic pressure on coal stockpile dominantly influences the flow inside stockpile. Velocity field in coal stockpile can be analysed and potentially hazard region inclinable to spontaneous ignition is localized.

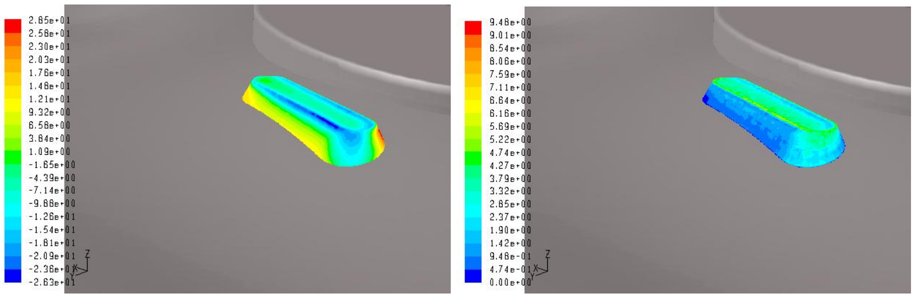

Fig. 7 Contour of static pressure (left) and velocity magnitude (right) on surface of coal stockpile, direction of wind South-West

6 CONCLUSIONS

The paper summarises the process of CFD simulation in atmospheric boundary layer with real topology of Earth's surface, which is necessary of solution of spontaneous ignition of coal matter in coal stockpile and dump. Flow of air in the surround of coal stockpile influences the results of the problem. Simulation with real atmospheric data is impossible in praxis. This is why the problem have to be simplify. Only cases with dominant direction of wind are solved. Dominant directions of wind are analyzed by means of meteorological data. Then simplified simulations with dominant directions of wind and stratification of atmosphere are solved. Additional simplification consists in mine buildings replace by porous domain with equal resistance coefficient. Results of these simulations are sources of boundary conditions for simulations with detail domain which includes coal stockpile with crack and definition of chemical reaction of low temperature oxidation and spontaneous ignition.

ACKNOWLEDGEMENT

Projekt byl realizován za finanční podpory ze státních prostředků prostřednictvím Grantové agentury České republiky. Registrační číslo projektu je GAČR 105/08/1414.

REFERENCES

[1] Michalcová, V.: Numerické modelování zatížení budov při kvazistatickém působení větru: Disertační práce Fakulta stavební, VŠB–TU Ostrava, Ostrava 2007. ISBN 978-80-248-1529-9

[2] Vesteeg, H. K., Malalasekera, W.: An Introduction to Computational Fluid Dynamics, Pearson Education Limited, England, 2007. ISBN 978-0-13-127498-3.

[3] Bojko, M., Kozubkova, M. Methodic of the mathematical modelling of the low-temperature oxidation of coal (spontaneous of coal) defined in porous zone. *Mechanical Engineering journal – Strojírenství.* KET SjF Košice, 2009, pp. 15-17, ISSN 1335-2938

[4] Fluent 6.3, Users Guide, Tutorial Guide