Numerical Analyses of Heat Transfer and Fluid Flow in Coal Depot and Mill

BRANIMIR PAVKOVIĆ*, ZORAN ČARIJA**, VELJKO KARUZA***
*Department of Thermodynamics and Energy Engineering
**Department of Fluid Mechanics and Computational Engineering
University in Rijeka - Faculty of Engineering
Vukovarska 58, 51000 Rijeka
*** Rijekaprojekt KONING
Moše Albahari 11a, 51000 Rijeka
CROATIA
http://www.riteh.hr

Abstract: - The paper presents CFD analyses performed during the design phase of a coal mill and a coal depot in a cement factory situated at eastern Adriatic coast. Analyses of the thermal distribution within the enclosed coal depot have been performed in order to achieve proper air distribution and to avoid increased temperatures which could lead to the coal self - ignition. Analyses of buoyancy forced air flow through the coal mill have been performed in order to find proper conditions for natural ventilation and avoid excessive air temperatures within the building. Simulation results have proven that both tasks could be solved with natural ventilation.

Key-Words: - heat transfer, buoyancy forced flow, computational fluid dynamics

1 Introduction

A cement factory situated at the eastern Adriatic coast, in the vicinity of Split has been reconstructed in order to meet increased demands for environmental protection. Among other, a reconstruction of the coal depot and coal mill has been performed. Closed coal storage has been provided in order to avoid spreading of the coal dust to the surroundings caused by wind. The coal mill is situated inside the building in order to reduce the noise and dust pollution. Such a solution could caus problems with increased temperature in closed buildings. Increased temperatures caused by solar radiation and equipment heat dissipation can be a threat for coal self - ignition at the depot, and also are dangerous for personell and control equipment within the coal mill.

Fig.1 Coal mill and a coal depot in the cement factory

Due to high heat rates which have to be removed from the internal spaces, the mechanical ventilation has been rejected as the possible solution for temperature control, and the natural ventilation conditions have been checked.

2 Problem Formulation

2.1 The coal mill

Heat sources within the coal mill are pipes for media transport, fans, fan electric motors, air - heaters and the mill with its electric motor. Pipes and equipment maintained at higher temperatures have been insulated, thus ensuring surface temperatures lower than 50°C, but electric motors and fans could not be insulated. Surface temperatures of the equipment are shown in Figure 2.

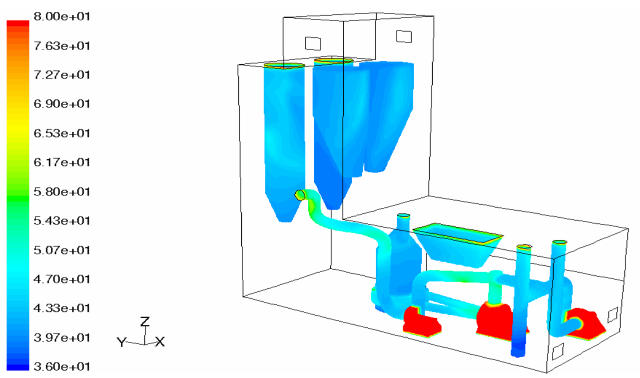

Fig.2 Surface temperatures of the equipment within the coal mill in the scale between 36°C and 80°C

All the equipment dissipates the heat, which causes buoyancy forced air flow. Solar energy gain has been considered as well. In the case of the coal mill with concrete walls, this gain has been found as insignificant when compared to internal heat sources. For properly placed ventilation openings at the building envelope, the temperature of the bulk air flow should not exceed 50^{o}C which has been set as the task of the research. In order to find proper positions and dimensions of ventilation openings, several arrangements were considered.

2.2 The coal depot

The dominant heat source in the case of the enclosed coal depot is the solar radiation which can lead to significant increase of internal temperature in the combination with increased summer air temperatures. Desired internal air bulk temperature in the vicinity of coal pile has been set to 50^{o}C as the limit which can give some guarantees against the self - ignition of the coal dust. Two cases have been analyzed, the first one refers to the possibility of ventilating through two doors which should be opened in the case of increased internal temperature (Figure 3) and the second one was the case with air vents along the entire building. In both cases openings for air outlet have been provided at the roof ridge of the building.

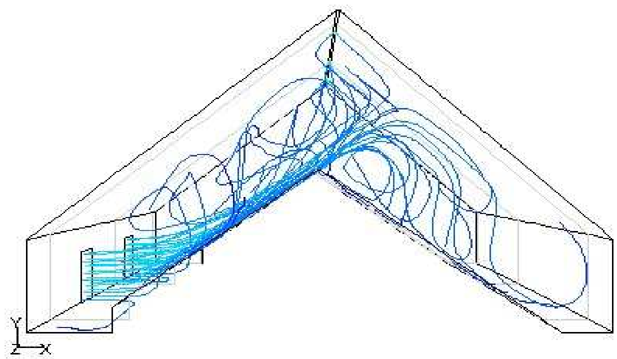

Fig.3 Coal depot ventilation by the door opening - flow paths

3 Problem Solution

3.1 Numerical modeling

Both buildings have been analyzed using CFD (Computational Fluid Dynamic). CFD is a standard procedure for simulation and analysis of fluid flow. In this process the fluid flow domain is divided into small volumes where governing equations are converted into algebraic equations, which are consequently solved numerically. Computational results strongly depend on applied mathematical model and numerical methods used for converting governing to algebraic equations.

A commercial computer code Fluent (version 6.2) has been utilized for the presented flow analysis. Fluent solves Reynolds averaged Navier-Stokes equations applied on finite volumes, where Reynold's stress is calculated from the applied turbulence models. Governing equations for predicting turbulent fluid flow and heat transfer are mass, momentum and energy conservation equations (1), (2) and (3)

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \left(\rho \vec{v}\right) = 0 , \qquad (1)$$

$$\frac{\partial}{\partial t}\left(\rho \vec{v}\right) + \nabla \cdot \left(\rho \vec{v}\vec{v}\right) = -\nabla p + \nabla \cdot \left(\bar{\bar{\tau}}\right) + \rho \vec{g} , \qquad (2)$$

$$\frac{\partial}{\partial t}\left(\rho E\right) + \nabla \cdot \left(\vec{v}\left(\rho E + p\right)\right) = \nabla \cdot \left(k_{eff} \nabla T + \left(\overline{\overline{\tau_{eff}}} \cdot \vec{v}\right)\right) + S_h , \qquad (3)$$

where E, T, S_h, v, p, μ and ρ are energy, temperature, heat source, velocity, pressure, dynamic viscosity and density.

Steady state fluid flow has been assumed for the fluid flow simulation. Fluent uses a control-volume-based technique to convert the governing equations to algebraic equations that can be solved numerically. Governing equations are solved sequentially (segregated solver). A variety of pressure-based algorithms are available in Fluent. For the present steady-state computations, the SIMPLE [1] algorithm has been adopted and second order upwind scheme has been used for convection terms discretization in the momentum and turbulent quantities transport equations. The resulting system of equations has been solved using an algebraic multigrid method for faster convergence.

Influence of applied turbulence model on calculated results has been investigated using standard k-ε model (SKE), and realizable k-ε model (RKE) [2] with standard wall functions. The major difference between those models is the method for calculation of turbulent viscosity and the way of defining generation and destruction terms in the turbulence dissipation rate equation. Standard k-ε model is mostly used in industrial applications today due to its robustness and satisfactory accuracy. Enhanced k-ε turbulence models (RKE) are better defined and are more consistent with the physics of turbulent flow. This turbulence models shown good performance for complicated flows with strong curvature, vortices and rotation, such as are in presented problems, and its results are going to be presented in this paper.

3.2 Mesh

Fluent CFD solver was applied for the fluid flow simulation using meshes with different number of finite volume cells (coal mill and coal depot). Meshes used in this work were created according to turbulence model requirements with each wall adjacent volume cell's centroid located within the log-law layer (30<Y+< 300). In order to satisfy this condition generated mesh was continuously corrected in near wall region (according to the fluid flow solution) until above conditions is not satisfied in the whole computational domain.

3.3 Boundary conditions

The surrounding air temperature has been assumed 36°C. Solar radiation on horizontal surface has been achieved from meteorological data for Split area, and simultaneous values for surfaces with different orientations have been calculated according to [3] and assumed as boundary values. Equal heat fluxes from boundary surfaces towards surroundings and the internal air have been assumed. Temperatures of boundary surfaces have been calculated separately from wall energy balances in preceding time step. Constant floor temperature of 25°C has been assumed. Coal pile surface has been assumed as adiabatic. Temperatures of the equipment inside the coal mill have been assumed constant according to technical specifications of thermal insulation, as well as process sheets and diagrams.

4 Results

4.1 The coal depot

Total air volume inside the coal depot is 40.000 m^3, while the coal pile volume is 12.000 m^3. For the first case, with 64 mm wide air vents placed above the ground level along the north and south wall of the building, and 180 mm wide air outlet along the entire ridge, 2-D model has been used. Results are shown in Figure 4. The difference between north and south side of the building caused by solar radiation is obvious. Higher temperatures are present in the upper part of the building, under the roof ridge.

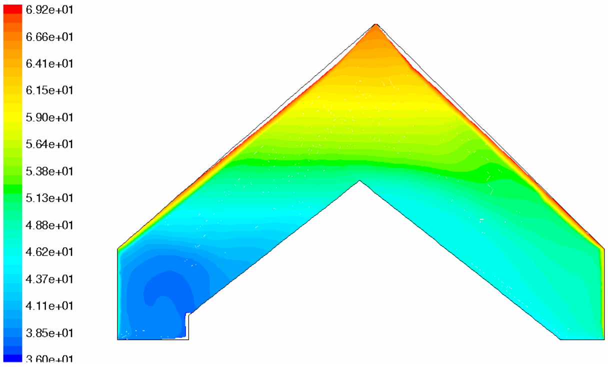

Fig.4 Temperature distribution inside the coal depot in the scale between 36°C and 69°C - results of 2-D model

For the second case, with the possibility of door opening during the extreme indoor temperatures and air outlet under the roof ridge, the 2-D model was not sufficient due to asymmetric geometry, and flow has been calculated using 3-D model.

Results of simulations are presented in Figures 5, 6 and 7. Figure 5 represents air - velocities for four characteristic cross sections of the building.

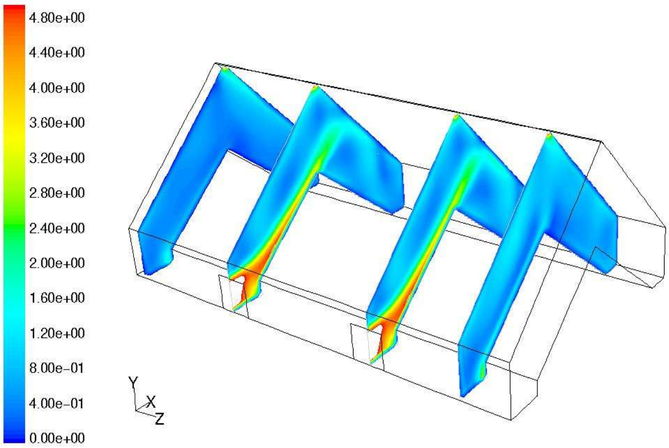

Fig.5 Velocity distribution inside the coal depot in the scale between 0 and 4,8 m/s - results of 3-D model

From figure 6, where temperature profiles are presented for the same four cross sections, the influence of better ventilation in the vicinity of doors is obvious.

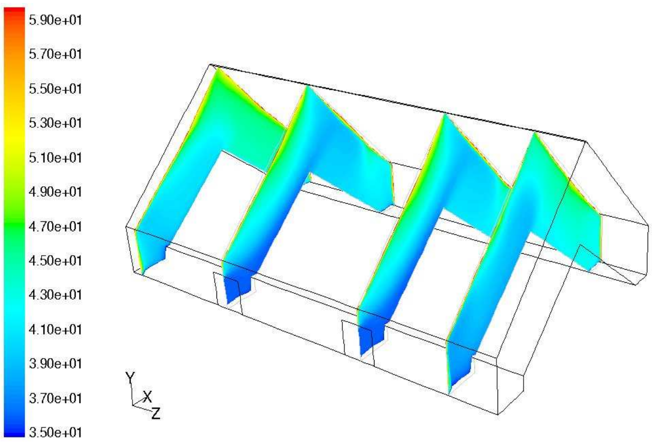

Fig.6 Temperature distribution inside the coal depot in the scale between 35°C and 59°C - results of 3-D model

Air temperatures in the boundary layer close to the coal pile are presented in figure 7. The air in the vicinity of the coal surface has been maintained at temperatures lower than requested 50°C, but poor air circulation over the south part of the coal pile is obvious and the proposed solution with opened doors has been abandoned.

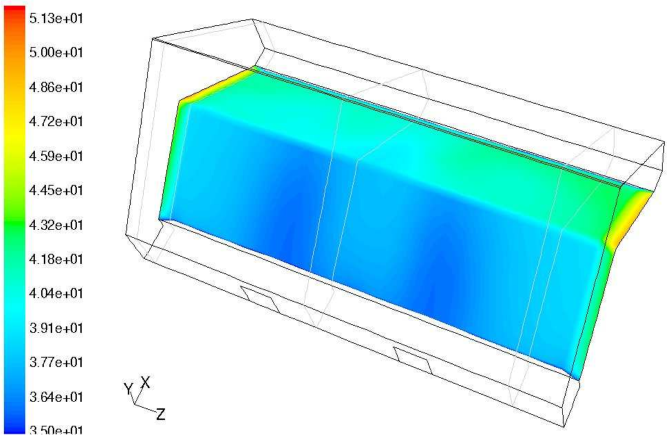

Fig.7 Temperature distribution in the boundary layer close to the coal pile - scale between 35°C and 55°C - results of 3-D model

4.2 The coal mill

Total volume of the coal mill is 13.500 m^3, while the air volume is 11.000 m^3. The rest of the volume is occupied by the equipment. Several positions of air vents have been considered and simulated. One satisfactory solution with inlet air vents' total cross area of 8 m^2 and outlet air vents' total cross area of 10 m^2 is presented in figures 8, 9 and 10, where temperature profiles are given for three different longitudinal cross sections. Path lines of the air are presented in figure 10 together with temperature distribution.

Total air volume flow induced by buoyancy is 190.000 m^3/h. Satisfactory bulk air temperatures are obvious for all cross sections.

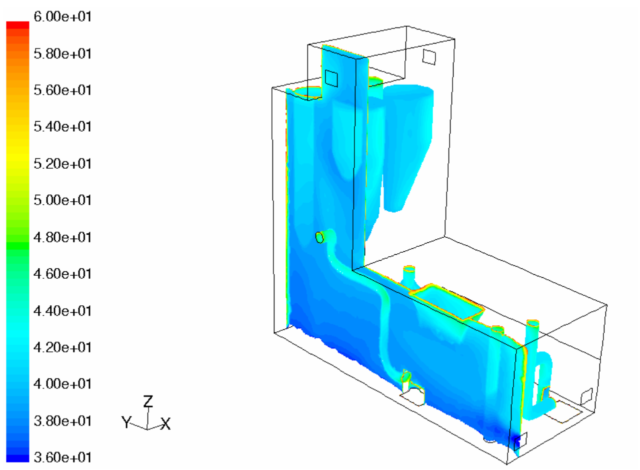

Fig.8 Temperature distribution inside the coal mill in the scale between 36°C and 60°C at $x = 2$ m - results of 3-D model

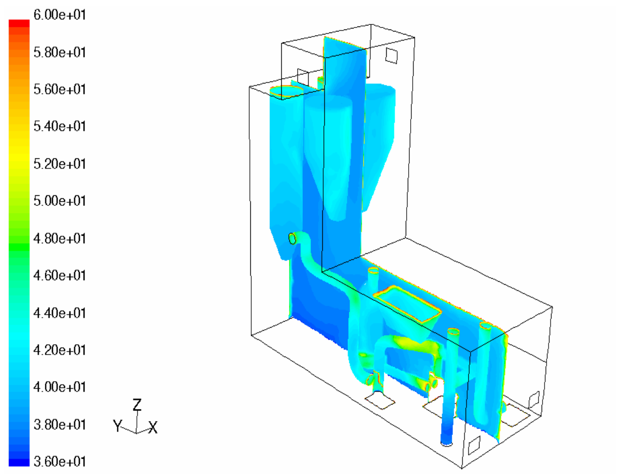

Fig.9 Temperature distribution inside the coal mill in the scale between 36°C and 60°C at $x = 7$ m - results of 3-D model

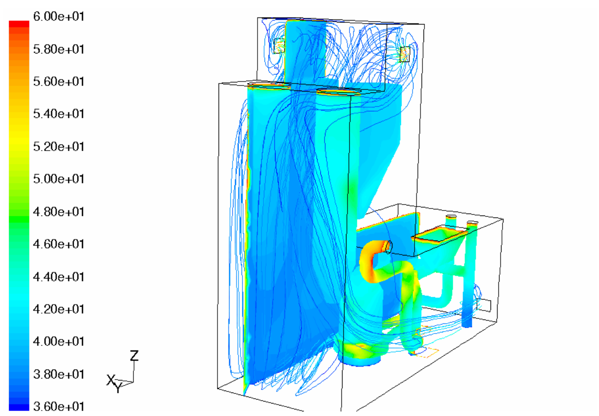

Fig.10 Air flow path - lines and temperature distribution inside the coal mill in the scale between 36°C and 60°C at $x = 12$ m - results of 3-D model

5 Conclusion

Computational fluid dynamic has been proven to be the powerful tool in solving the buoyancy - driven air flow within considered buildings, thus helping to solve the problem of their natural ventilation and to maintain temperatures within desired limits.

References:

[1] Patankar S. V., *Numerical Heat Transfer and Fluid Flow*, Hemisphere, Washington, D.C., 1980.

[2] Fluent 6.2 *User manual*, Fluent.Inc, 2006.

[3] Pavković, B., Medica, V.; *Optimization of solar consumption water heating systems based on average monthly data*, Proc. 3rd International Congress SITHOK, Maribor 1998., 243 – 251