UDK: 532/533:519.876.5:621.18
COSATI: 20-04, 20-13, 12-01

Comparison of Numerical and Experimental Results for Multiphase Flow in Duct System of Thermal Power Plant

Mirko Kozić[1)]
Slavica Ristić[2)]
Mirjana Puharić[2)]
Boris Katavić[2)]
Mirjana Prvulović[2)]

This paper presents the numerical simulation results of the multiphase flow in the ventilation mills of Kostolac B power plant, where blinds and centrifugal separators with adjustable blade angles are used. The numerical simulation was performed using the mixture model in the Euler-Euler approach of commercial ANSYS FLUENT software package. The gas mixture distribution and the velocity of sand particles in the mill and duct system are given in the paper. The results of the numerical simulations are compared with the measurements for both types of separators. Based on the numerical results, the critical locations exposed to the most intense wear in the mill and duct system are determined.

Key words: **fluid dynamics, fluid flow, multiphase flow, ventilation system, ventilation mill, gas mixture, numerical methods, numerical simulation, power plant.**

Introduction

MULTIDISCIPLINARY research of ventilation mills of lignite power plants includes a variety of theoretical, numerical, empirical and experimental methods of flow research[1-24]. A numerical simulation of the flow is the most economical, fastest, and a very reliable method of analyzing complex issues of multiphase flow and its optimization. Every contribution to the energy efficiency of thermal power plants, energy saving and environmental protection represents a significant result.

The characters of multiphase flow recirculation gas, coal powder, sand and other materials are directly related to the efficiency of the ventilation mill [1-3]. The construction and geometry of the ventilation mill and the mixture channel along with the wear process of vital parts significantly affect the energy efficiency of the plants. In Kostolac B power plant eight fan mills are used. The odd numbered mills have channels with blinds and also channels with built-in centrifugal separators.

This paper presents the results of the multiphase flow numerical simulation in the mill and duct system consisting of recirculation gas and sand particles. The distribution and velocity of sand are very important to be determined because sand particles cause strong wear of the grinding wheel elements, mill walls, separators and decrease both time period between two repairs and life span [1, 9, 25-26]. The mill duct systems with the blinds and the centrifugal separator configurations are considered. The results of numerical simulation are compared with the measurements performed in the mill thermal plant [2, 24].

Ventilation mill

In Kostolac B power plant, coal is ground in the ventilation mills. The system includes eight ventilation mills of EVT N 270.45 type, with a nominal capacity of 76 t/h of coal. Each mill is directly connected to the burner system consisting of four levels; two bottom, main burners of coal powder mixture and two upper or secondary burners. Fig. 1 shows photos of the mill-duct system in Kostolac B power plant. The technical characteristics of the ventilation mill are as follows [1-3]: the impeller diameter (working wheel diameter): D=3600 mm; the revolution per minute in the interval 420–500 ^{o}/min, the average revolution per minute w=480 ^{o}/min [25-26].

The test results presented in literature [1-3] show that in the mills with blind separators, better distribution of coal powder is achieved compared to the centrifugal separator, giving 82% and 18% for the main and secondary burners, respectively. The appropriate distribution for the ventilation mills with the centrifugal separator is 62:38% of lignite powder for the position of the blades of about 20°. This distribution of coal powder has a major influence on the combustion process. The gas mixture distribution is also more regular for blinds with 52% for the main burners, opposite to 45% with the centrifugal separator. This contributes to improving the transport properties and combustion process.

The aforementioned reconstructions resulted in 70:30% distribution of the coal powder and 60:40% of the gas mixture on the main and secondary burners, in accordance with the requirements of the supplier. The quality of milling

[1)] Military Technical Institute (VTI), Ratka Resanovića 1, 11132 Belgrade, SERBIA
[2)] Institute GOŠA d.o.o., Milana Rakića 35, 11000 Belgrade, SERBIA

was improved slightly, so that the rests on the sieves R90 and R1000 were 65-70% and below 10%, respectively. But the tests, performed after reconstruction, have shown that the distribution 70:30% of the coal powder is not optimal and proper combustion could not be obtained. The possibility of additional reconstructions was considered.

Figure 1. Ventilation mill in Kostolac B power plant

Together with the larger lignite particles from the separator of the mill, a significant amount of the cooled products returns, therefore reduces the ventilation effect of the mill and increases the energy consumption for grinding. Abrasive particles, enriched with minerals of higher density then coal, significantly increase the wear of the mill working elements, reducing the period between two repairs.

The velocity field of the sand particles is investigated. The kinetic energy of these particles is important for the wear of the mill vital parts. The ventilation effect depends a little on the separation system and under normal conditions is about 200,000 m^3/h.. Fig.2 shows some parts of the ventilation mill in reconstruction [25-26]

a) b)

c

Figure 2. Reconstruction of the ventilation mill:; a-wall plate coating, b-mill wheel and c-impact plate

Numerical methods

There is a wide range of software for numerical simulation. Numerical simulations represent an important alternative to laboratory tests especially when investigations on complex thermal power plants are needed [8-23]. Numerical methods are an essential tool in engineering analysis used in all branches of science and technology for different kind of simulations.

The commercial ANSYS FLUENT software is based on the finite volume method. There are two approaches for the numerical simulation of the multiphase flow known as Euler-Lagrange and Euler-Euler approach. In the first approach, the primary phase is treated as continuum by solving the time-averaged Navier-Stokes equations. The behavior of the dispersed phases is obtained by following a large number of the particles, through the calculated primary phase flow field. Particle trajectories are calculated in the given intervals during the primary phase flow calculations. The dispersed and primary phases can exchange mass, momentum and energy. The basic assumption in this model is that the volume fraction of the dispersed, secondary phase is below 10%, although the mass dispersed phase can be even greater than the mass of the primary phase.

The different phases in the Euler-Euler approach are considered as inter-penetrating continua, thus enabling introduction of the volume fractions as continuous functions of time and space. The sum for all phase volume fractions in each computational cell is equal to one. The conservation laws are applied to each phase in order to obtain a set of equations that is similar for all phases. Constitutive relations obtained from empirical information must be added to close the set of equations. In the Euler-Euler approach, there are three models of multiphase flow: the model of volume of fluid (VOF), the mixture model and the Eulerian model.

The mixture model is a simplified multiphase model that can be used in modeling flows where the phases move at different velocities using the concept of slip velocities. The existence of local equilibrium at small length scales is assumed. This model can include n phases, where the equations of continuity, momentum and energy conservation are solved for the mixture, while the volume fraction equations are determined for each of the secondary phases. Algebraic equations are used in solving the relative velocities. This model enables a selection of granular secondary phases and can be used as a good substitution for the full Eulerian multiphase model, in cases with wide size distribution of the solid phase or when the inter phase laws are unknown.

The Eulerian model is the most complex of all models of multiphase flow in the used software. In this model the additional equations of continuity and momentum exchange for each phase are solved separately. Phases can be liquids, gases and solid particles in any combination. The Euler's method of determining the flow field is used for both the primary and secondary phase. Coupling between phases is included through the pressure and the coefficient of intermediate exchange, and the manner in which these gain depends on the types of phases involved in the flow. Thus, for the flow with solid fractions, the pressure and coefficient of intermediate exchange are obtained by the kinetic theory. The number of secondary phases in the Euler model is practically limited by the available memory and behavior of solution convergence.

Numerical flow simulation

Mixture model

This paper presents the results of the numerical simulation obtained using the mixture model in the Euler-Euler approach. The volume fractions in the control volume can have any value between 0 and 1. In addition, the model allows phases to move with different velocities introducing slip velocity between them.

The continuity equation for the mixture is

$$\frac{\partial}{\partial t}\rho_m + \nabla\cdot\left(\rho_m \vec{v}_m\right) = 0 \tag{1}$$

where $\vec{v}_m$ is the mass-averaged velocity

$$\vec{v}_m = \frac{\sum_{k=1}^{n}\alpha_k \rho_k \vec{v}_k}{\rho_m} \tag{2}$$

and ρ_m is the mixture density

$$\rho_m = \sum_{k=1}^{n}\alpha_k \rho_k \tag{3}$$

whereas α_k is the volume fraction of the phase k.

The equation of the momentum exchange is obtained by summing the individual ones for all phases,

$$\begin{aligned}&\frac{\partial}{\partial t}\left(\rho_m \vec{v}_m\right) + \nabla\cdot\left(\rho_m \vec{v}_m \vec{v}_m\right) = \\ &-\nabla p + \left[\mu_m\left(\nabla\vec{v}_m + \nabla\vec{v}_m^T\right)\right] + \rho_m \vec{g} + \vec{F} + \\ &\nabla\cdot\left(\sum_{k=1}^{n}\alpha_k \rho_k \vec{v}_{dr,r}\vec{v}_{dr,r}\right)\end{aligned} \tag{4}$$

where n is the number of phases, $\vec{F}$ is the body force, and μ_m is the viscosity of the mixture.

The viscosity of the mixture is given as

$$\mu_m = \sum_{k=1}^{n}\alpha_k \mu_k \tag{5}$$

while $\vec{v}_{dr,k}$ is the drift velocity for the secondary phase k

$$\vec{v}_{dr,k} = \vec{v}_k - \vec{v}_m \tag{6}$$

The energy equation for the mixture is also obtained by summing all phases,

$$\begin{aligned}&\frac{\partial}{\partial t}\sum_{k=1}^{n}\left(\alpha_k \rho_k E_k\right) + \nabla\cdot\sum_{k=1}^{n}\left(\alpha_k \vec{v}_k\left(\rho_k E_k + p\right)\right) = \\ &= \nabla\cdot\left(k_{eff}\nabla T\right) + S_E\end{aligned} \tag{7}$$

where k_{eff} is the effective conductivity. The first term on the right side represents energy transfer due to conduction, while S_E includes any other volumetric heat sources.

The relative velocity is defined as the velocity of a secondary phase p relative to the velocity of the primary phase q,

$$\vec{v}_{pq} = \vec{v}_p - \vec{v}_q \tag{8}$$

The mass fraction for any phase k is defined as

$$c_k = \frac{\alpha_k \rho_k}{\rho_m} \tag{9}$$

so the drift velocity and the relative velocity are connected by

$$\vec{v}_{dr,p} = \vec{v}_{pq} - \sum_{k=1}^{n} c_k \vec{v}_{qk} \tag{10}$$

In FLUENT's mixture model an algebraic formulation is used in calculating relative velocity, based on the assumption of the local equilibrium between the phases over a short spatial length scale.

The form of the relative velocity most often used is given in [1] as

$$\vec{v}_{pq} = \frac{\tau_p}{f_{drag}}\frac{\left(\rho_p - \rho_m\right)}{\rho_p}\vec{a} \tag{11}$$

where τ_p is the particle relaxation time defined as

$$\tau_p = \frac{\rho_p d_p^2}{18\cdot\mu_q} \tag{12}$$

where d_p is the diameter of the particles of the secondary phase p, $\vec{a}$ is the secondary phase particle's acceleration.

The default drag function taken from [2], is:

$$f_{drag} = \begin{cases}1 + 0{,}15\,\mathrm{Re}^{0{,}687}\cdots\mathrm{Re}\le 1000\\ 0{,}0183\,\mathrm{Re}\ \ \cdots\cdots\mathrm{Re}\ge 1000\end{cases} \tag{13}$$

while the acceleration is of the form

$$\vec{a} = \vec{g} - \left(\vec{v}_m\cdot\nabla\right)\vec{v}_m - \frac{\partial\vec{v}_m}{\partial t} \tag{14}$$

In the simplest algebraic model, the acceleration of the particle is given by gravity and centrifugal force, whereas the relaxation time is modified to take into account the presence of other particles. The volume fraction for the secondary phase p can be obtained from the continuity equation for this phase,

$$\begin{aligned}&\frac{\partial}{\partial t}\left(\alpha_p \rho_p\right) + \nabla\cdot\left(\alpha_p \rho_p \vec{v}_m\right) = \\ &= -\nabla\cdot\left(\alpha_p \rho_p \vec{v}_{dr,p}\right) + \sum_{q=1}^{n}\left(\dot{m}_{qp} - \dot{m}_{pq}\right)\end{aligned} \tag{18}$$

The concentration of particles is important in the calculation of the effective viscosity for the mixture, and in this case the granular viscosity is usually used. This viscosity arises from particles momentum exchange due to translation and collision. The shear viscosity is obtained as a sum of the collisional $\mu_{s,col}$ [3, 4 and the kinetic part $\mu_{s,kin}$ [4]:

$$\mu_s = \mu_{s,col} + \mu_{s,kin} \tag{19}$$

The effects of the turbulent fluctuations of velocity scalar quantities are modeled using various types of closure models. The number of terms to be modeled in multiphase flows is large compared to single-phase flow. The FLUENT provides the k-ε and the Reynolds-Stress model. There are three options for the *k*-*ε* model: mixture turbulence model, dispersed turbulence model and turbulence model for each phase. The RSM turbulence model options are the mixture turbulence model and the dispersed turbulence model.

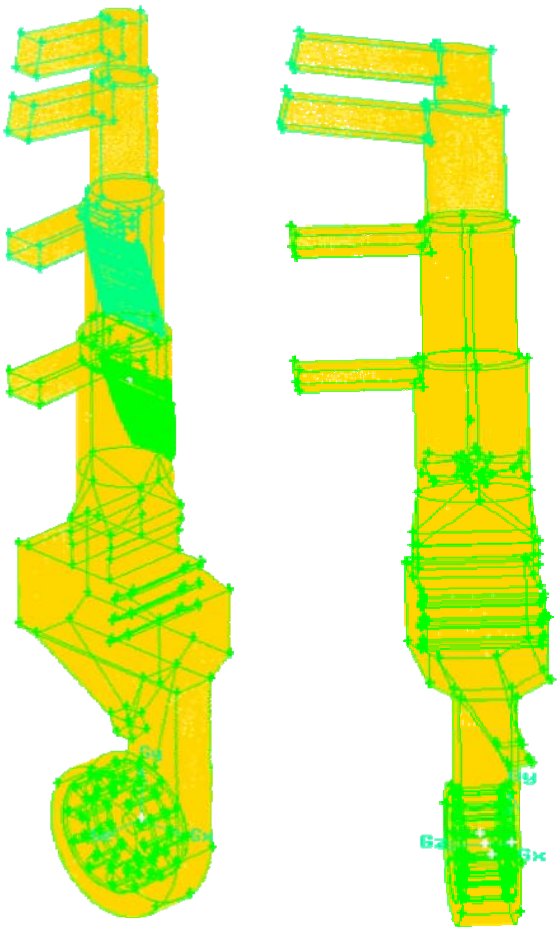
Numerical simulation of flow in the ventilation mill and duct system

The first step of the numerical modeling of multiphase flow in the ventilation mill and the mixture channel is geometry preparation and mesh generation. A detailed description is given in references [25-26].

The second step is the preparation of the input data for a solver. It includes the definition of the general flow model, multiphase model, phases and their interactions, viscous model and the turbulence model and boundary conditions. In addition, the initialization of the flow field is accomplished, along with the convergence solution monitoring and post-processing.

The smallest details from the original geometry have been omitted because the mesh in such a model had too many elements. Figs.3a and 3b show the model geometry and the volume grid in the whole numerical domain for the mill with the blinds (a) and the centrifugal separator (b).

For the purposes of the simulation, an unstructured tetrahedral grid is generated with 2996772 volume elements and 706444 surface elements.



a b

Figure 3. Volume mesh in the whole numerical domain of the ventilation mill with, a- blinds, and b-centrifugal separator

The input data for the numerical simulation of the multiphase flow in the ventilation mill with blinds were determined based on the measurements conducted in 2008. The measurements were performed on mills 23 and 25 of the boiler B2. The appropriate parameters for the numerical simulation are calculated as the mean value for each mill. Two values of the input data for the numerical simulation are given, where the first is related to the mill with blinds, and the second to the one with centrifugal separator:

- volume flow rate of recirculation gases:

$$\dot{V}_{sr} = 95.87\,\mathrm{m}^3/\mathrm{s}\,, \quad \dot{V}_{sr} = 107.66\,m^3/s$$

- mass flow rate of lignite:

$$\dot{m}_{sr} = 16.72\ \mathrm{kg/s}\,, \quad \dot{m}_{sr} = 17.97\,kg/s$$

- moisture content in pulverized coal:

$$w_p = 5.68\,\%\,, \quad w_p = 5.68\,\%$$

- mass flow rate of pulverized coal:

$$\dot{m}_{pulv.coal} = 7.28\,kg/s.\,, \quad \dot{m}_{ug.prah} = 6.37\ \ \mathrm{kg/s}$$

- volume fractions of the secondary phases:

$$\alpha_{pul.coal} = 5.11 \cdot 10^{-5}\,, \quad \alpha_{pul.coal} = 5.65 \cdot 10^{-5}$$

$$\alpha_{sand} = 1.67 \cdot 10^{-5}\,, \quad \alpha_{sand} = 1.81 \cdot 10^{-5}$$

$$\alpha_{wat.vap.} = 0.135\,, \quad \alpha_{wat.vap.} = 0.129$$

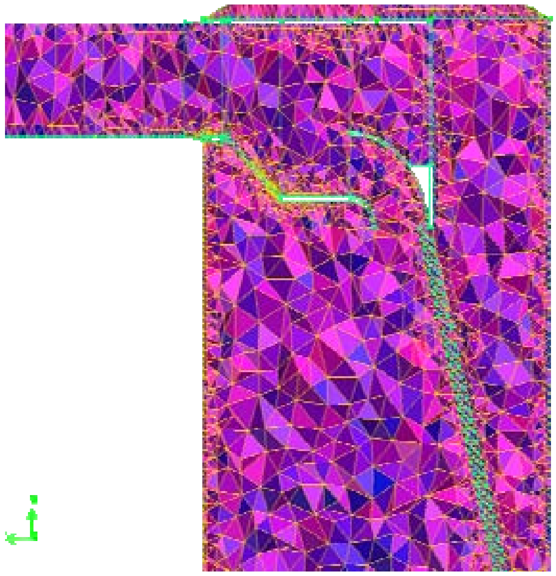
The volume fractions of the secondary phases are very small, giving dilute mixture. The lignite powder is modeled as a mono disperse granular secondary phase, with particles diameter equal to 150 microns. The particle diameter of sand is taken to be 300μm. The particle weight and drag are accounted for. For collisions between particles of the granular phases, the value of restitution coefficient 0.9 is chosen. The default k-ε mixture turbulence model is used in modeling turbulence fluctuations.

The standard no-slip boundary condition is applied at all walls including the mill impeller that rotates with 495 rpm. Its rotation is modeled with the multiple reference frames (MRF) option in the software. The walls of the mixture channels are well insulated, so the adiabatic thermal boundary condition is applied. At all exits the value of static pressure is defined. The velocity is defined at the mill entry in such a way that the volume flow rate of the recirculation gases be satisfied. The first order accurate numerical discretization is used, because the calculation with the second-order schemes is unstable.

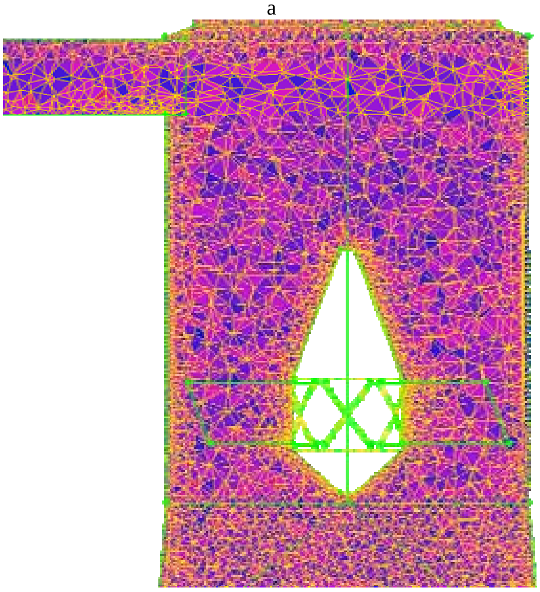
In the analysis of the results it should be taken into account that the numerical simulations have some limitations. The first type of the constraint is related to the complexity of the physical models incorporated in the used software and the possibility of obtaining relevant results. This especially holds for turbulence models, multiphase flows and combustion models. In the real ventilation mill coal is milled. However, the software ANSYS FLUENT 12 belongs to CFD codes in which the process of milling can not be modeled. Therefore, in the numerical simulation the solution is obtained as if the mixture of recirculation gases, pulverized coal and sand entered the ventilation mill.

Another type of restriction is a very complex geometry which includes the impeller mill, the mill housing, a large number of close-packed blinds, as well as a complex geometry of the centrifugal separator and burners.

In Figs.4a and 4b the vertical plane section through the volume mesh for the blinds and centrifugal separator is shown.



a



b

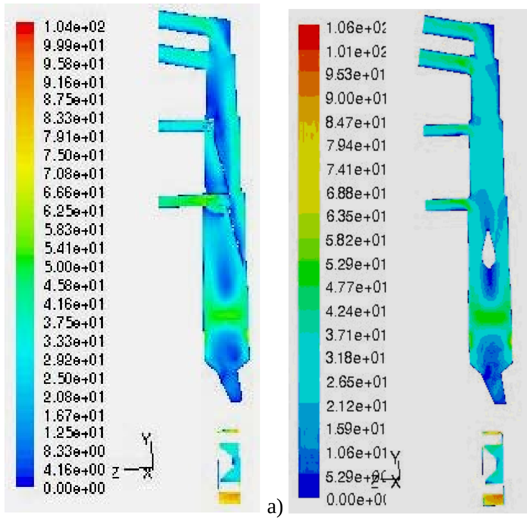
Figure 4. Cross section of the volume mesh for the ventilation mill with a-blinds, and b-centrifugal separator

Results of the numerical simulation and discussion

The results of the numerical simulation for both types of the separators are presented using tables and figures. The gas mixture distributions obtained numerically and experimentally at the main and secondary burners are compared in Table 1. In Table 2 numerical and experimental values of the gas mixture average velocity for each burner are given.

The comparisons of the measurements and the numerical simulation show very good agreement as to the distribution of the gas mixture with differences up to 7% for both configurations. The numerical and experimental values of the gas mixture velocity at the main lower burner and at the secondary upper burner in the configuration with the blinds are very close. However, at the secondary lower and the main upper burner the difference is about 25%. Better agreement of the measured and numerical results is obtained for the mill configuration with the centrifugal separator, where the largest difference equal to 18% is obtained for the velocity of the gas mixture at the main upper burner.

In Fig.5 the velocity field of the gas mixture in a vertical plane passing through the rotation axis of the mill with the blinds (a) and with the centrifugal separator (b) can be seen. The highest velocity of the gas mixture appears in the mill due to its rotation, while in the duct system the velocity magnitude is up to 50 m/s. It is observed that there is an increase of velocity in the transition zones from vertical to horizontal pipes leading to the main and secondary burners, and also the zones of very low velocity, especially a stagnation one in front of the body of the centrifugal separator.



a) b)

Figure 5. The velocity field of the gas mixture in a vertical plane passing through the rotation axis of the mill, a-with the blinds and b-centrifugal separator

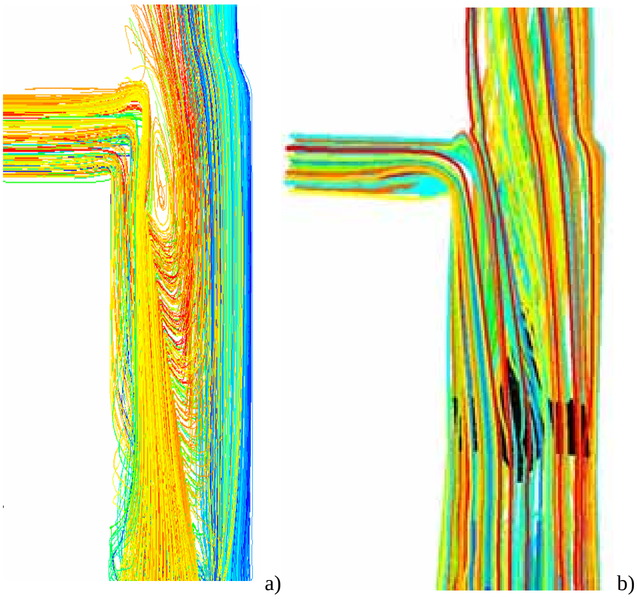
Table 1. Gas mixture distribution

	Gas mixture distribution (blinds)		Gas mixture distribution (blades of centrifugal separator at angle 20°)	
	measurements [16]	numerical simulation	measurements	numerical simulation
main burners	52 %	55 %	45%	48%
secondary burners	48 %	45 %	55%	52%

Table 2. Average velocity of gas mixture

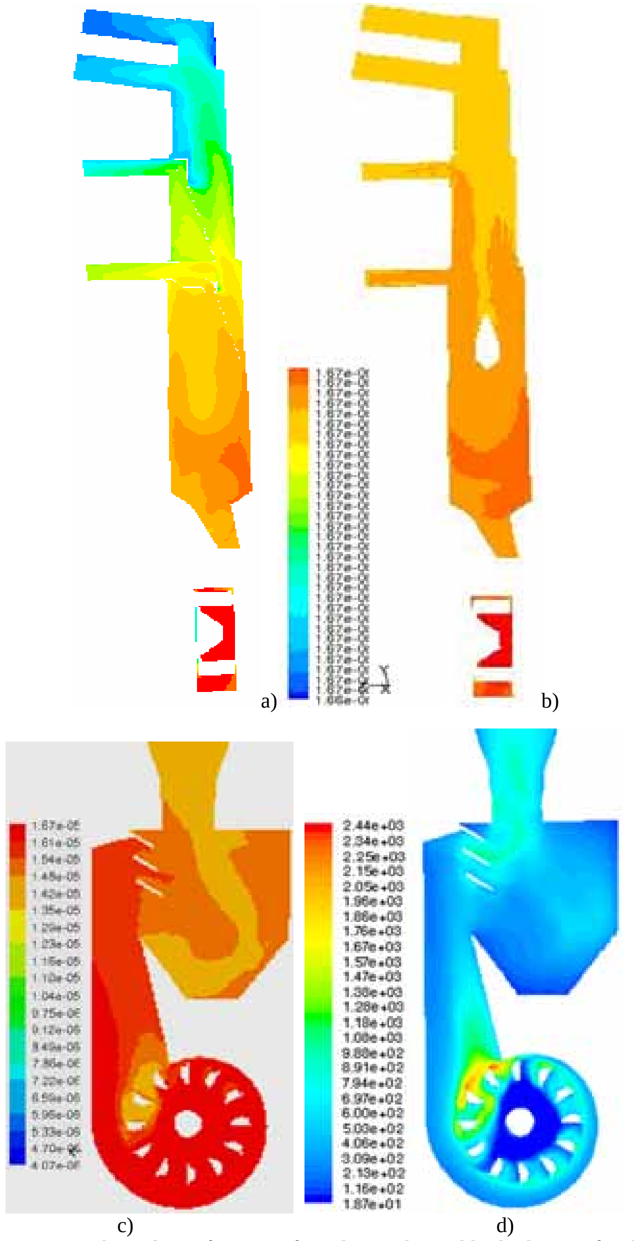
	Average velocity (m/s) (blinds)		Average velocity (m/s) (blades of centrifugal separator at angle 20°)	
	measurements [17]	numerical simulation	measurements [17]	numerical simulation
main lower burner	30.3	30.7	32.2	30.0
main upper burner	27.0	21.3	22.0	27.1
secondary lower burner	17.3	22.4	24.2	24.6
secondary upper burner	22.8	22.0	24.3	24.9

In Fig.6 the path lines of the gas mixture at the lower blinds and the lower main burner (a), and around the centrifugal separator with the blades at 20^{o} (b) are shown. Fig.7 depicts the volume fraction of sand for both types of the separators, (a), (b), (c), and the gas mixture turbulence in the mill and duct (d). The largest turbulence occurs at the transition from the mill to the duct, where reversal of the flow exists due to the mill rotation.



a) b)

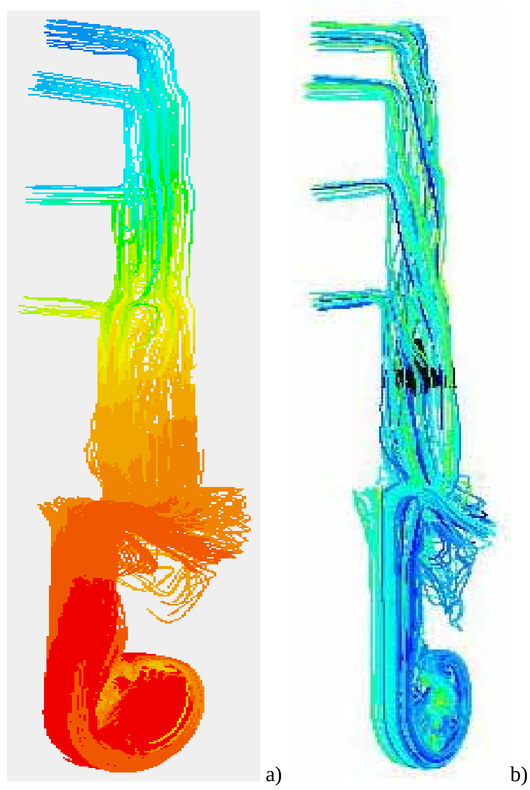
Figure 6. The path lines; a-the gas mixture at the main burner and the lower blinds and b-the centrifugal separator with the blades at 20^o



a) b) c) d)

Figure 7. The volume fraction of sand particles; a-blinds, b-centrifugal separator, c-zoom of the ventilation mill in the plane normal to the rotation axis, d-turbulence of the gas mixture in the mill

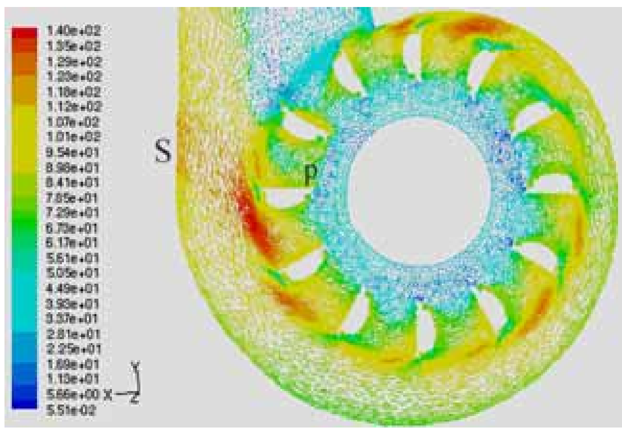
The path lines of sand from the mill entry to the exits of the main and secondary burners are shown in Fig.8, for the configuration with the blinds (a) and the centrifugal separator (b). The volume fractions of the granular secondary phases that include lignite powder and sand are very low, meaning that the mixture is dilute. In such a case the path lines of the mixture and the ones of the granular phases will be practically the same.



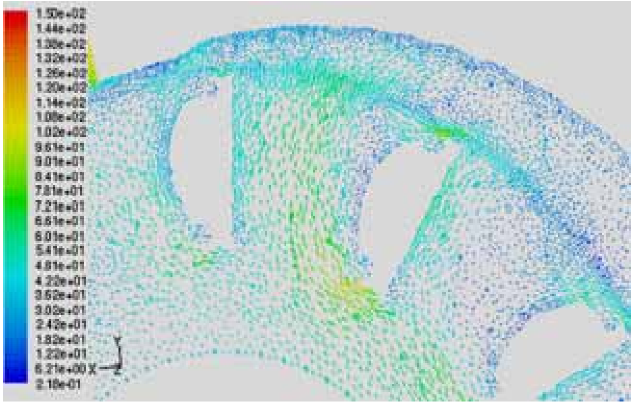
a) b)

Figure 8. The path lines of sand from mill to exits of the main and secondary burners, a-blinds, and b-centrifugal separator

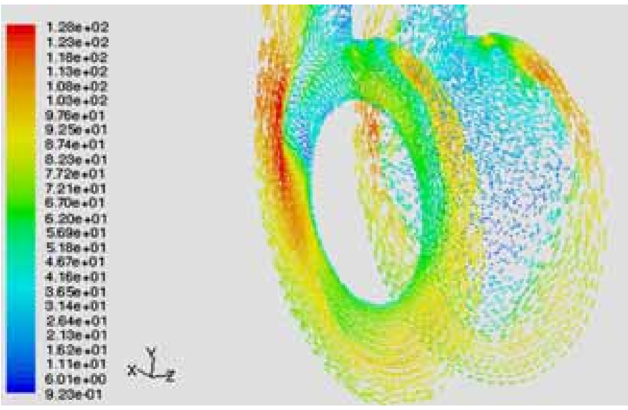
The absolute velocity vectors of the gas mixture and sand in the mill are shown in Figs.9a, and 9c. It can be seen that the maximum velocity magnitude on the side walls of the mill is about 120 m/s. Such high velocity causes intensive wear of the mill walls and protective plates. In Fig.9b the relative velocity vectors of sand particles around the impact plates of the grinding wheel are shown. The wheel and the abraded impact plates of the ventilation mill in reconstruction are presented in Figs.2b and 2c.



a)



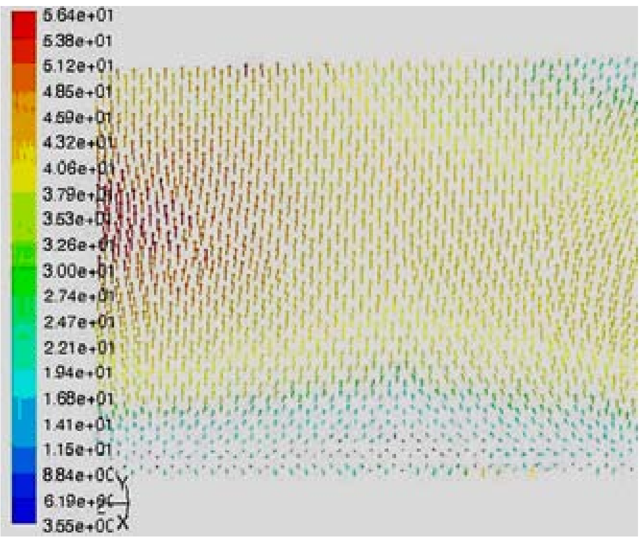
b



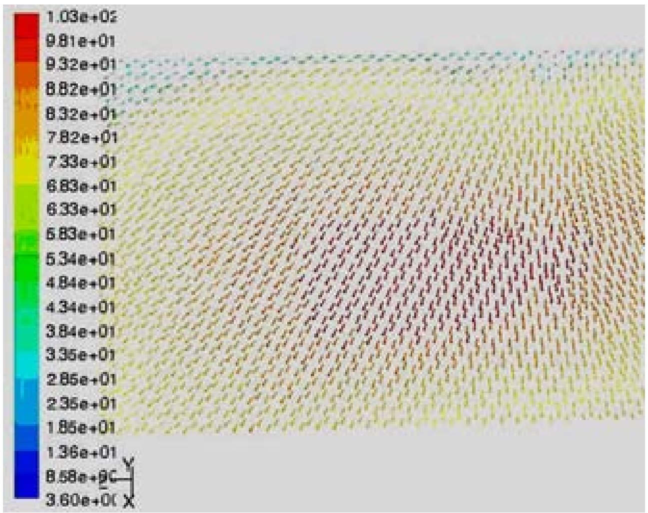
c

Figure 9. a-absolute velocity vectors of the gas mixture in the mill, b-relative velocity vectors of sand particles around the impact plates, c-absolute velocity vectors of sand particles on the side walls of the mill

Due to the grinding wheel rotation, the velocity vectors of sand particles relative to the impact plates, especially to their front surfaces, are of great importance because of pronounced wear at these places. In Fig.10a and 10b the relative velocity vectors of sand are shown on the front surface of the impact plate at 30^o after the most upper position, and 90^o after the lowest position, respectively. The maximum relative velocity magnitude of sand particles, which is about 100 m/s, causes damage to the impact plate front surfaces. It has to be emphasized that the directions of the relative velocity vectors obtained by the numerical simulation and the noticed damage (Fig.2) on the front surfaces of the impact plates are in a very good agreement.



a)



b)

Figure 10. a-Relative velocity vectors of sand on the front surface of the impact plate at 30^o after the most upper position, b-relative velocity vectors of sand on the upper surface of the impact plate at 60^o after the lowest position

Conclusion

The results obtained by the numerical simulation of the multiphase flow in the ventilation mill EVT N 270.45 of Kostolac B power plant clearly show that the CFD methods provide a detailed flow pattern in the complex plant. The numerical simulation was performed for two kinds of separators in the duct system, i.e. blinds and centrifugal separator. The paper presents results concerning the distribution of the gas mixture at the main and secondary burners, the value of the gas mixture velocity for each burner, and the velocity vectors of the gas mixture and sand particles in the mill.

The distribution of the gas mixture for the main and secondary burners, and the gas mixture average velocity at the main and secondary burners obtained numerically are in a good agreement with the measurements.

It is shown in the simulation that the high absolute velocity magnitude of the gas mixture occurs on the side walls of the mill. It causes intensive wear of the mill walls and the protective plates observed in the operation. The maximum velocity magnitude of sand particles relative to the rotating grinding wheel, which is about 100 m/s, causes damage to the impact plates. The directions of the relative velocity vectors obtained numerically and real damage on the front surfaces of the impact plates are in a very good agreement.

Acknowledgments

This paper is the result of the research within the project TR 19205, financially supported by the Ministry of Science and Technology of Serbia, PD TE – KO Kostolac and Messer Tehnogas.

References

[1] PERKOVIĆ,B., MAZURKIJEVIČ,A., TARASEK,V., STEVIĆ,LJ.: *Reconstruction, and realization of the projected modrnization of power block B2 in the TE Costal*, Termotehnika, 2004, Vol.XXX, No.1, str.57-81.

[2] *Thermo investigation and analysis of boiler plant blocks B1 and B2 in Costal TE*, PD TENT Ltd. Production-technical sector, 2007, 2008 (internal study in Serbian),

[3] GULIČ,M.: *Calculation of fan mill*, Beograd, 1982,

[4] KUAN,B.T., YANG,W., SOLNORDAL,C.: *CFD simulation and experimental validation of dilute particulate turbulent flows in* 90º *duct bend*, Third International conference on CFD in the minerals and process industries CSIRO, Melbourne, Australia, 2003; 10/12 dec.Proceedings, 531-536,

[5] KUAN,B.T.: *CFD simulation of dilute gas-solid two-phase flows with different solid size distributions in a curved* 90º *duct bend*, 2005, ANZIAM J. C744-C763,

[6] LOPEZ,A.L., OBERKAMPF,W.L.: *Numerical simulation of laminar flow in curved duct*, SAND 95-0012C, conf-950130-6,

[7] TUCAKOVIĆ,D., ŽIVANOVIĆ,T., STEVANOVIĆ,V., BELOŠEVIĆ,S., GALIĆ,R.: *A computer code for the prediction of mill gases and hot air distribution between burners' sections at the utility boiler*, Applied Thermal Engineering, 28, 2008, pp. 2178–2186,

[8] BELOŠEVIĆ,S., SIJERČIĆ,M., TUCAKOVIĆ,D., CRNOMARKOVIĆ,N.: *A numerical study of a utility boiler tangentially-fired furnace under different operating conditions*, Fuel, 2008; 87: pp.3331–3338,

[9] LIMING,Y., SMITH,A.C.: *Numerical study on effects of coal properties on spontaneous heating in longwall gob areas*, Fuel, 87, 2008, pp.3409–3419,

[10] BHASTER,C.: Numerical simulation of turbulent flow in complex geometries used in power plants, Advances in Engineering software, 2002, 33, pp.71-83,

[11] MANNINEN,M., TAIVASSALO,V., HJERTAGER,B.H.: *On mathematical models of turbulent combustion with special emphasis on soot formation and combustion*, 16th Symp. On Combustion, The Combustion Institute, 1976,

[12] SCHILLER,L., NAUMANN,Z., Ver.Deutch,Z.: Ing. 1935, pp.77-318.

[13] GIDASPOW,D., BEZBURUAH,R., DING,J.: *Hydrodynamics of Circulating Fluidized Beds, Kinetic Theory Approach*, Proceedings of the 7th Engineering Foundation Conference on Fluidization, 1992, pp.75-82,

[14] SYAMLAL,M., ROGERS,W., O'BRIEN,T.J.: *MFIX Documentation: Volume I, Theory Guide*, National Technical Information Service, Springfield, VA, 1993,

[15] DREW,D.A., LAHEY,R.T.: *In Particulate Two-Phase Flow*, 509-566, Butterworth-Heinemann, Boston, 1993,

[16] BAKER,B.J., WAINWRIGHT,T.E.: *Studies in Molecular Dynamics II: Behaviour of a Small Number of Elastic Spheres*, J.Chem.Phys., 1990, pp.33-439.

[17] CHAPMAN,S., COWLING,T.G.: *The Mathematical Theory of Non-Uniform Gases*, Cambridge University Press, Cambridge, England, 3rd edition,1990,

[18] DING,J., GIDASPOW,D.: *A Bubbling Fluidization Model Using Kinetic Theory of Granular Flow*, AICHE J., 36, 1990, pp.523-538,

[19] MORSI,S.A., ALEXANDER,A.J.: *An Investigation of Particle Trajectories in Two-Phase Flow System*, J. Fluid Mech., September 26, 1972, Proceedings, 55, 1972, pp.193-208,

[20] SYAMLAL,M., O'BRIEN,T.J.: *Computer Simulation of Bubbles in a Fluidized Bed*, AICHE Symp. Series., 85, 1989, pp. 22-31,

[21] WEN,C.Y., YU,Y.H.: *Mechanics of Fluidization*, Chem.Eng.Prog.Symp. Series, 62, 1966, pp.100-111,

[22] RANZ,W.E., MARSHALL,W.R.,Jr.: *Evaporation from Drops, Part I*, Chem.Eng.Prog, 48, 1952,pp. 141-146,

[23] CSANADY,G.T.: *Turbulent Diffusion of Heavy Particles in the Atmosphere*, J.Atmos. Science, 20, 1963, pp.201-208,

[24] SIMONIN,C., VIOLLET,P.L.: *Predictions of an Oxygen Droplet Pulverization in a Compressible Subsonic Coflowing Hydrogen Flow*, Numerical Methods for Multiphase Flows, FED91, 1990, pp.65-82,

[25] KATAVIĆ,B., KOZIĆ,M., RISTIĆ,S., PUHARIĆ,M.: *Numerička simulacija strujanja u ventilacionom mlinu termoelektrane Drmno-Kostolac- konfiguracija sa žaluzinama*, elaborat, Institut Goša, 2009,

[26] KOZIĆ,M., KATAVIĆ,B., RISTIĆ,S., PUHARIĆ,M.: *Numerička simulacija strujanja u ventilacionom mlinu termoelektrane Drmno-Kostolac-konfiguracija sa centrifugalnim separatorom*, elaborat, Institut Goša, 2010,

Received: 08.11.2010.

Poređenje numeričkih i eksperimentalnih rezultata za multifazno strujanje u sistemu kanala termoelektrane

U radu su prikazani rezultati numeričke simulacije višefaznog strujanja u ventilacionim mlinovima termoelektrane Kostolac B u kojima se koriste žaluzine i centrifugalni razdvajači sa podešavajućim uglom lopatica. Numerička simulacija je vršena korišćenjem modela mešavine u pristupu Ojler-Ojler komercijalnog softverskog paketa ANSYS FLUENT. U radu je data raspodela gasne mešavine i brzine čestica peska. Dobijeni rezultati numeričke simulacije su upoređeni sa merenjima za oba tipa razdvajača. Na osnovu numeričkih rezultata, određena su kritična mesta u mlinu i kanalu aerosmeše na kojima dolazi do najintenzivnijeg habanja.

Ključne reči: **dinaika fluida, strujanje fluida, višefazno strujanje, ventilacion sistem, ventilacioni mlin, gasna meštavina, numeričke metode, numerička simulacija, termoelektrana.**

Сравнение расчетных и эьных результат многофазного течения в системе канала термоэлектростанции

В настоящей работе показаны цифровые моделирования многофазного потока в проветривающих мельницах термоэлектростанции Костолац Б, в которых пользуются жалюзами и центробежными разделителями с регулируемым углом лопасток. Цифровое моделирование проведено при пользовании модели смеси в подходе Эйлер-Эйлер коммерческого пакета программного обеспечения ANSYS FLUENT. В этой работе показано расспределение газа и скорости частиц песка. Полученные результаты цифрового моделирования сопоставлены с измерениями для обоих типов разделителей. На обосновании цифровых результатов, определены критические места в мельнице и в трубопроводе аэросмеси, на которых происходит самое напряжённое (интенсивное) изнашивание.

Ключевые слова: **динамика жидкости, поток жидкости, многофазный поток, проветривающая система, проветривающая мельница, газовая смесь, цифровые методы, цифровое моделирование, термоэлектростанция.**

Une comparaison des résultats numériques et expérimentaux pour multiphase l'ecoulement dans un systéme des canaux centrale

Ce papier présente les résultats de la simulation numérique du courant polyphasé dans les moulins de ventilation du centrale thermique Kostolac B où sont utilisés les séparateurs centrifuges aux lames à l'angle réglable. La simulation numérique a été effectuée à l'aide du modèle de mélange dans l'approche de Euler-Euler du progiciel commercial ANSYS FLUENT. Dans ce travail on a donné la distribution du mélange de gaz et la vélocité des particules de sable. Les résultats obtenus de la simulation numérique étaient comparés avec les mesurages pour les deux types de séparateurs. A la base des résultats numériques on a déterminé les endroits critiques dans le moulin et dans la conduite de mélange aérien où se produisent les usures les plus intenses.

Mots clés: **dynamique des fluides , courant des fluides, courant polyphasé, système de ventilation, moulin de ventilation, mélange de gaz, méthodes numériques, simulation numérique, centrale thermique.**