NUMERICAL SIMULATION OF THE INFLUENCE OF STATIONARY LOUVER AND COAL PARTICLE SIZE ON DISTRIBUTION OF PULVERIZED COAL TO THE FEED DUCTS OF A POWER PLANT BURNER

by

Goran ŽIVKOVIĆ, Stevan NEMODA,
Predrag STEFANOVIĆ, and Predrag RADOVANOVIĆ

Original scientific paper
UDC: 662.62:532.517.4:66.011
DOI: 10.2298/TSCI0904079Ž

One of the key requirements related to successful utilization of plasma technology as an oil-free backup system for coal ignition and combustion stabilization in power plant boilers is provision of properly regulated pulverized coal distribution to the feed ducts leading the fuel mixture to a burner. Proper regulation of coal distribution is deemed essential for achieving an adequate pulverized coal concentration in the zone where thermal plasma is being introduced. The said can be efficiently achieved by installation of stationary louver in the coal-air mixing duct ahead of the feed ducts of a burner. The paper addresses numerical simulation of a two-phase flow of air-pulverized coal mixture in the mixing ducts, analysing the effects of particle size distribution on pulverized coal distribution to the burner feed ducts. Numerical simulation was performed using the FLUENT 6.3 commercial code and related poly-dispersed flow module, based on the PSI-CELL approach. Numerical experiments have been performed assuming a mono-dispersed solid phase with particle diameter ranging from 45 μm to 1200 μm. Distance between the louver blades and the resulting effect on the flow profile was analysed as well. Results obtained indicate that the size of coal particles considerably influence the overall solid phase distribution. While fine particles, with diameters at the lower end of the above specified range, almost fully follow the streamlines of the continuous phase, coarser particles, which hit the louver blades, deflect towards the thermal plasma zone. In this manner, a desired phase concentration in the considered zone can be reached. For the said reason, installation of stationary louver have been deemed a very efficient way to induce phase separation, primarily due to more pronounced impact of the installed louver on discrete phase flow then the impact on the flow of the continuous phase.

Key words: *two-phase flow, pulverized coal, turbulent flow, louver, computational fluid dinamics*

Introduction

Instead of traditional systems where oil is being used as a backup fuel to support the coal ignition and combustion, in the Unit A of the thermal power plant (TPP) Nikola Tesla, low temperature plasma generators have been installed in the burner feed duct of mills no. 2 and 5 of the boiler no. 1. The main advantage provided by the said assembly is the use of pulverized coal instead of imported oil. In order for the specified system to provide required combustion support and operate properly it is necessary to achieve appropriate mass concentration of pulverised coal in the coal-air mixture flowing through the burner feed ducts. The required coal concentration equals 0.23. The specified coal-air mixture, after being formed in the mills, is distributed to eight feed ducts supplying the burner with the fuel, four of them equipped with electric arc plasma generators. Each plasma generator has an output of approximately 100 kW and is able to locally heat up the air up to 5000 K.

In a case when flow regulation is not provided, the stream of pulverized coal is evenly distributed to each burner feed duct, resulting in coal concentration in the mixture which is lower than the above mentioned critical value. It is therefore deemed necessary to introduce a device for pulverized coal redistribution, to be positioned in front of the inlets of the burner feed ducts. The said device would increase coal concentration in the feed ducts equipped with plasma generators. Classifier-type devices for pulverized coal redistribution have already been used but have not been able to provide desired level of phase separation. The paper analyses possible solution to the above mentioned problem provided through installation of additional pulverized coal redistribution system based on the system of stationary louvers. The specified solution was proposed due to the good results achieved by installation of similar systems in many power plants in Europe. The basic advantages of stationary louvers utilisation for pulverized coal redistribution are: (1) relatively low additional pressure drop of about 100 Pa and consequently small impact on the proper mill operation and (2) easy control of pulverized coal distribution resulting from the fact that some louvers may be designed as non-stationary, *i. e.* with an adjustable inclination angle.

The main objective of the research presented in the paper was to examine the efficiency of louver utilization as an instrument for redistribution of pulverized coal to the burner feed ducts. Parameters determining the flow pattern of coal-air mixture are the following: length and inclination angle of louver blades, spacing between the blades, and the angle of stationary louvers in the feed duct. In order to achieve optimal results with the addition of the louver system it was necessary to perform detail analysis of the effects imposed by certain parameters, *i. e.* to determine flow profile of the coal-air mixture and the coal and air flow distribution to the burner feed duct. Numerical flow model and corresponding computer simulation represent an optimal tool for accomplishing the specified tasks. Numerical modelling and computer simulation enable easy variation of the most important flow parameters and analysis of resulting effects, leading to the final conclusion on the optimal values of specific parameters. For this reason, for many years numerical simulation of two-phase dispersed flow has been considered to be the most appropriate tool for the specified purpose. Experimental verification of numerical computation is usually performed only for a few characteristic cases. If computational results show good agreement with data obtained from the related measurements, the numerical model is deemed verified. Besides, the considered approach significantly reduces associated research costs. In accordance with the above said, a thorough numerical analysis have been performed, analysing the impact of louvers installation on the flow pattern of the considered mixture, as well as the impact of a distance between the louver blades on the flow distribution to the burner feed ducts. Such analysis is specially important for the mills installed in TPP Nikola Tesla since geometry of the mills, as well as their operating parameters, vary significantly from one mill to another. The said means that a solution determined to be efficient in the case of one particular mill can not be automatically implemented to the others, *i. e.* not without a preliminary analysis of the resulting effects. Otherwise, besides not being able to provide results desired, louver installation may even cause a complete malfunction of the mill, induced by the large pressure drop through the louvers. Analysis of the effects imposed by all relevant parameters of louver geometry on flow characteristics has not been displayed in the paper, but was presented in a technical report [1, 9, 10]. Data presented in the Technical report have shown that the said parameters could be crucial in establishing a particular flow profile and provision of required phase separation. Thus, the paper only presents a portion of comprehensive numerical research carried out for the purpose of considering the effects of louvers installation on distribution of pulverized coal to the burner feed ducts. The paper addresses the effects of particle size distribution and the number of louver blades, which are deemed the most important parameters determining the discrete-phase flow pattern.

Two-phase (pulverized coal – air mixture) flow modelling

There is no doubt that development of original models and respective numerical codes represent the best way to obtain appropriate numerical solution for a particular engineering problem. The said approach is deemed to be the most reliable and the most accurate. Many examples of such two-phase models can be found in the literature. However, the said approach is rather expensive and the entire process is long-lasting. On the other hand, commercial computational fluid dynamics (CFD) codes have recently achieved such a high level of efficiency and accuracy that their use in the research activities worldwide has demonstrated that the ones can be successfully used for simulation of high-complexity flows. Such flows, for example, develop in the duct behind the mill which is equipped with the system of stationary louvers. It is of course necessary for the user of such code to be provided with a possibility to "intervene" in the code, *i. e*. to adjust a general model to the particular problem in question. In the commercial software FLUENT, version 6.18, this could be easily achieved using the "user-defined functions". The software also contains an integrated modulus for computation of poly-dispersed mixture flows. For the said reasons the specified software has been selected to be used for flow simulation in the research presented.

A physical model developed in this modulus incorporates all main features of models described in the reference documents [2-5]. The model is based on Lagrangian approach of particle tracking, with phase coupling handled by the PSI-CELL model and stochastic particle motion model which has taken into account all relevant forces (drag force, lift force, Magnus force *etc*.). Coal particles were assumed to be spherical. Effects of turbulence on particle dispersion were predicted using the model of turbulent eddies in the continuous phase. Described numerical modelling resulted in defined coal particle trajectories, with specified particle position and velocity at every instant. The following represents the basic equation of particle motion:

$$\frac{\mathrm{d}\vec{\mathrm{u}}_{\mathrm{p}}}{\mathrm{d}t} \quad F_{\mathrm{D}}|\vec{\mathrm{u}} \quad \vec{\mathrm{u}}_{\mathrm{p}}| \quad \frac{\vec{\mathrm{g}}(\rho_{\mathrm{p}} \quad \rho)}{\rho_{\mathrm{p}}} \quad \vec{F} \tag{1}$$

where $F_{\mathrm{D}}|\vec{\mathrm{u}} \quad \vec{\mathrm{u}}_{\mathrm{p}}|$ represents a drag force, with:

$$F_{\mathrm{D}} \quad \frac{18\mu}{\rho_{\mathrm{p}} d_{\mathrm{p}}^{2}} \frac{C_{\mathrm{D}}\,\mathrm{Re}}{24} \tag{2}$$

where

$$\mathrm{Re} \quad \frac{\rho d_{\mathrm{p}}|\vec{\mathrm{u}} \quad \vec{\mathrm{u}}_{\mathrm{p}}|}{\mu} \tag{3}$$

Drag coefficient, C_{D}, was calculated using the following expression:

$$C_{\mathrm{D}} \quad a_1 \quad \frac{a_2}{\mathrm{Re}} \quad \frac{a_3}{\mathrm{Re}^2} \tag{4}$$

where coefficients a_1, a_2, and a_3 are a function of the Re and were taken from the observations of Morsi *et al.* [6].

Equation (1) incorporates an additional force $\vec{F}$ that in some cases may be deemed important. The specified equation term includes Saffman lift force, additional mass force, and additional force due to the pressure gradient in the fluid.

The Saffman lift force equation was obtained from the work of Li *et al.* [7] and represents a generalisation of the expression provided by Saffman:

$$\vec{F} \quad \frac{2K\nu^{0.5}\rho \mathrm{d}_{ij}}{\rho_\mathrm{p} \mathrm{d}_\mathrm{p} (\mathrm{d}_\mathrm{lk} \mathrm{d}_\mathrm{kl})^{0.25}} (\vec{\mathrm{u}} \quad \vec{\mathrm{u}}_\mathrm{p}) \tag{5}$$

where $K = 2.59$ and d_{ij} is a deformation tensor.

Additional mass force is obtained from the following relation:

$$\vec{F} \quad \frac{1}{2}\frac{\rho}{\rho_\mathrm{p}}\frac{\mathrm{d}}{\mathrm{d}t}(\vec{\mathrm{u}} \quad \vec{\mathrm{u}}_\mathrm{p}) \tag{6}$$

Additional lift force due to the pressure gradient in the fluid is obtained from the following equation:

$$\vec{F} \quad \frac{\rho}{\rho_\mathrm{p}}\vec{\mathrm{u}}_\mathrm{p} \quad \vec{\mathrm{u}} \tag{7}$$

The trajectory equations are solved by stepwise integration of the eq. (1) over discrete time steps, which yield the particle velocity in every point along the trajectory. Integration of the eq. (1) denotes the integration of three scalar equations (one for each direction). Assuming that the term in the equation containing the body forces remains constant over each time interval (this assumption is justified by the fact that these additional forces are of much lower intensity than the drag force or the influence of turbulent fluctuation of the continuous phase velocity) the following discretised equation is obtained:

$$\frac{\mathrm{d}\vec{\mathrm{u}}_\mathrm{p}}{\mathrm{d}t} \quad \frac{1}{\tau_p}(\vec{\mathrm{u}} \quad \vec{\mathrm{u}}_\mathrm{p}) \tag{8}$$

where τ_p is particle relaxation time. Equation (8) was solved using a trapezoidal numerical scheme.

Turbulent particle dispersion was modelled using a stochastic tracking approach. Based on the specified approach, turbulent particles dispersion is predicted by integrating the trajectory equation of individual particles, whereby using the instantaneous fluid velocity $\bar{u} \quad u$ along the particle path.

Task formulation

Geometry of the flow domain is shown in fig. 1a. Close to the side walls two plates have been introduced, aimed to prevent the particle agglomeration near the walls. Plasma generators were placed in four burner feed ducts on the right-hand side of the domain depicted, also equipped with blades for duct closing or opening. During the entire computational period the blades were held in a half-open position, as indicated in the figure. Louvers were positioned in front of the four left-hand ducts not containing plasma generators (in accordance with the basic idea to decrease coal concentration in plasma-free ducts and in the same time increase the concentration in feed ducts equipped with plasma generators), across the entire width of the duct. The louver blades were 120 mm long and 10 mm thick. The inclination angle toward the horizontal plane was 20°. The paper presents results obtained with louvers comprising 5 and 6 blades. Since the duct width is constant and equals approximately 1.5 m, a 2-D flow domain calculation has been performed. A part of the numerical grid obtained is shown in fig. 1b. As seen in the figure, the grid close to the louver blades is finer than the grid in the straight duct. Utilization of finer grid was necessary due to the intensive turbulent effects in the considered region. It is also noticeable that transition from finer to the coarser grid was made gradually. The total number of control volumes equalled 38000.

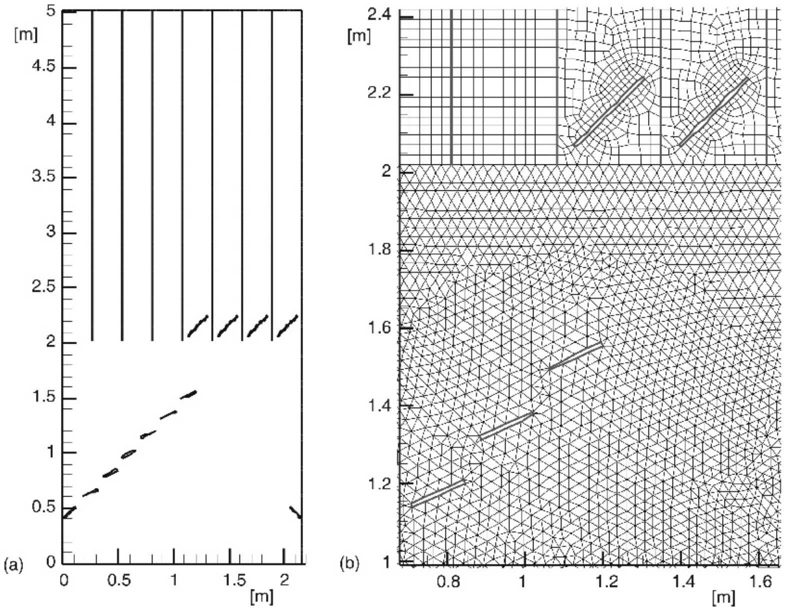

Figure 1. (a) Geometry of the flow domain; (b) Numerical grid in a section of the flow domain

Parcels (computational particles representing a large number of real particles of the same characteristics) entered the flow domain through the entire duct width, starting from 120 starting points uniformly distributed across the entrance section. Time step between two successive insertions was $5 \cdot 10^{-4}$ s, with the time period of each insertion of 2s (in fact, 4s in accordance with a comment provided below). In this manner, discrete phase velocity and concentration fields were determined based on statistical data processing of information obtained from 480000 parcels. The specified number of parcels included in the statistical analysis was deemed high enough to ensure representative statistical results. In fact, during the numerical simulation performed much higher number of parcels has been monitored, but the data obtained from the additional parcels had not been statistically processed. The said was mainly due to the requirement related to the steady-state conditions that needed to be reached in both phases, meaning that data recording started only after the parcels had filled the entire flow domain. With such large number of processed parcels and steady-state conditions reached in both phases, it was possible to use a non-stationary (Lagrangian) discrete phase model for steady-state simulation. In order to obtain desired statistical data, a special user-defined function (subroutines in the original code which provides the user with a possibility to intervene in the code) was defined.

Initial conditions for the computation performed were obtained from the measurement of mass flow rates of both coal-air mixture and pulverized coal, as well as the mixture temperature. The specified measurements were conducted in 2007 on the mill no. 5 installed in the Unit B1 of the TPP Nikola Tesla. The said measurements represented only a portion of the overall

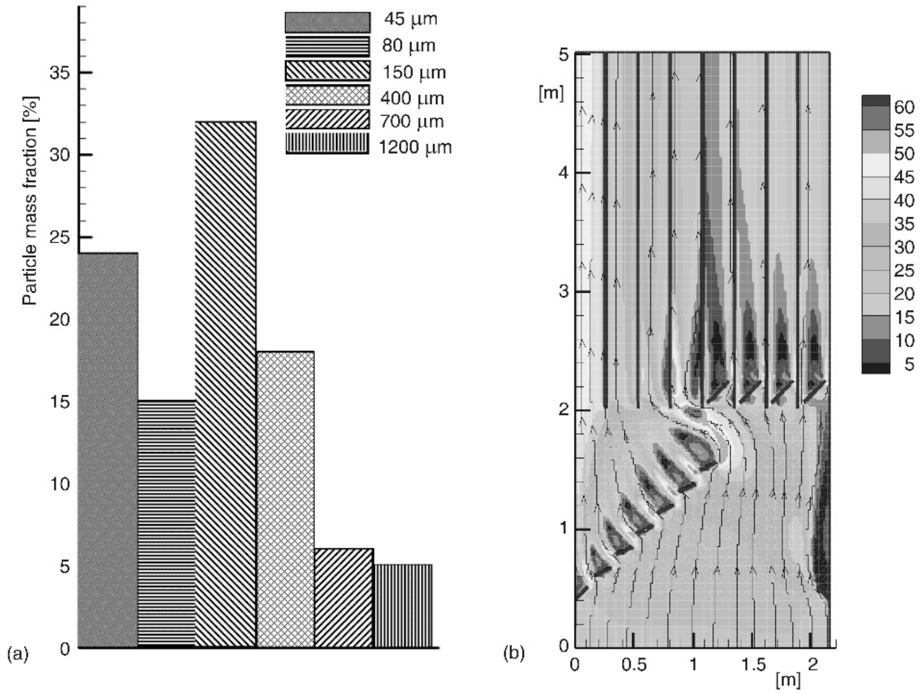

Figure 2. (a) Particle size distribution, % by weight; (b) Velocity profiles for the geometry with six louver blades

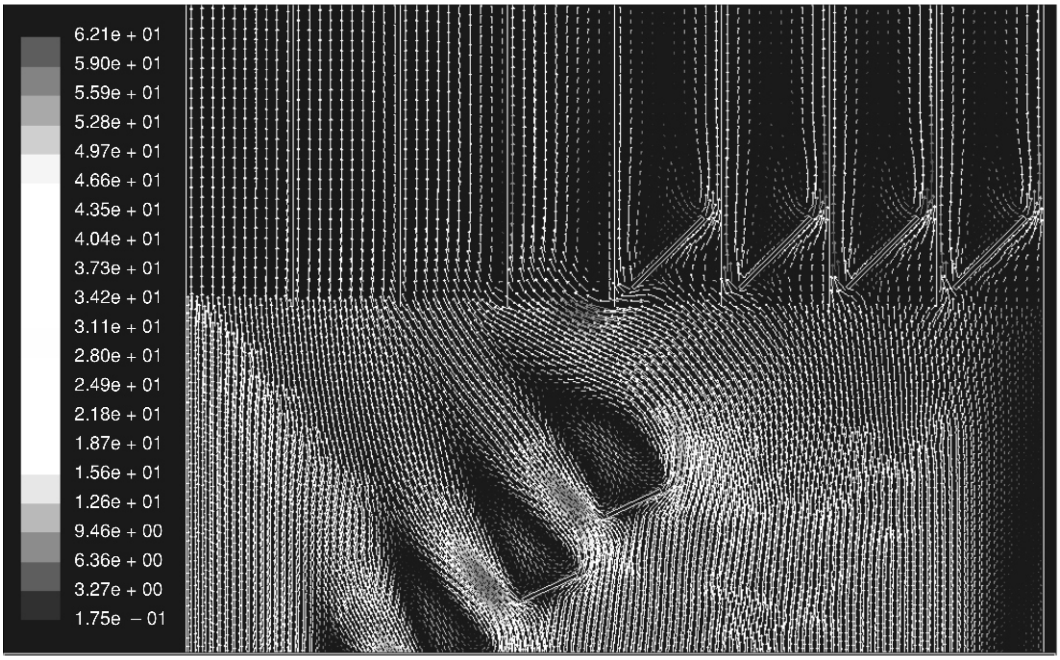

Figure 3. Flow pattern in the vicinity of the louver blades as indicated by velocity vectors

measurements carried out by Vinča Institute of Nuclear Sciences (Laboratory for Thermal Engineering and Energy) with an aim to examine the operation of five mills and a boiler installed in the Unit B1 of TPP Nikola Tesla. The main goal of the measurements was to get an insight into the power plant operation following the refurbishment of the unit, in order to optimise the plant performance. Measurements included an estimation of mill capacities, fineness of pulverization, coal moisture, pressure drop, flow rate of pulverised coal distributed to each burner, O_2 concentration in recirculation duct, *etc*.

It was assumed that the coal-air mixture was characterised by a uniform velocity of 24.8 m/s at the inlet of the flow domain and that the temperature of the mixture equalled 180 °C. Based on the particle size analysis, six characteristic coal particle sizes (45 m, 80 m, 150 m, 400 m, 700 m, 1.200 m diameters) were selected. Associated particle size distribution in the poly-dispersed mixture of pulverized coal is shown in fig. 2a.

Results of the numerical simulation

A characteristic flow velocity of the continuous phase (for the case with six louver blades installed) is shown in fig. 2b. In order to provide a closer insight into the considered flow pattern a few characteristic streamlines were also presented. Flow pattern in the continuous phase developed close to the louver blades, as indicated by velocity vectors, is presented in fig. 3.

Figures 4 and 5 show a parcel distribution in the flow domain at one instant of time under steady-state conditions for each of the six the above mentioned particle sizes and for five and six louver blades installed, respectively.

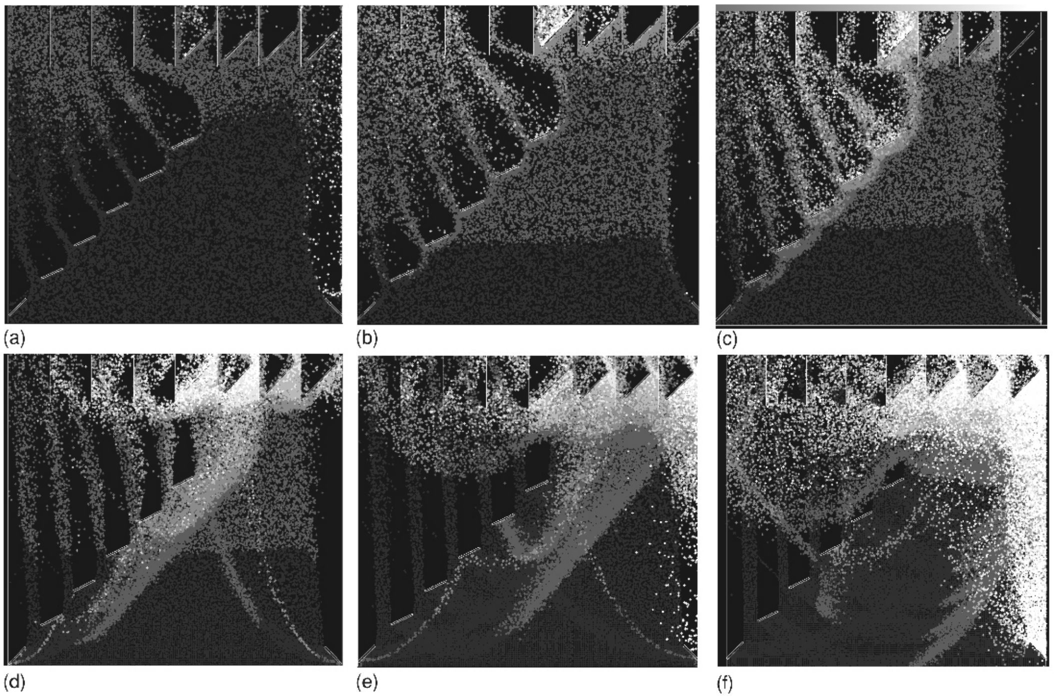

Figure 4. Steady-state particle distribution in the flow domain at one instant of time for 5 louver blades installed: (a) *d* = 45 μm; (b) *d* = 80 μm; (c) *d* = 150 μm; (d) *d* = 400 μm; (e) *d* = 700 μm; (f) *d* = 1200 μm

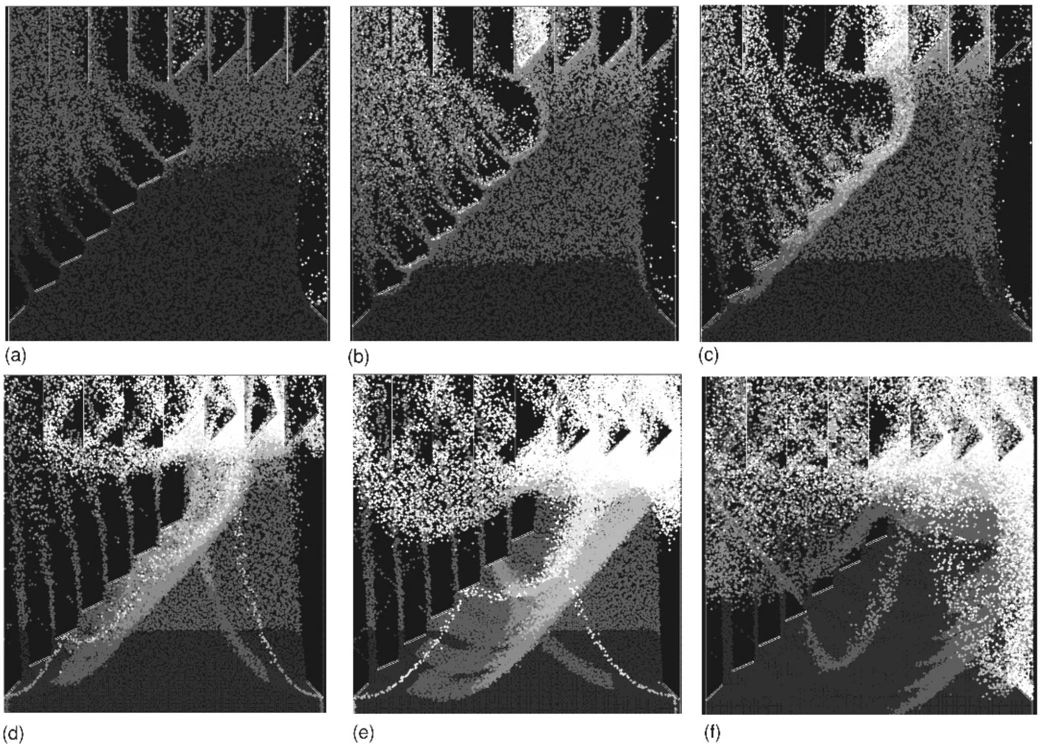

Figure 5. Steady-state particle distribution in the flow domain at one instant of time for 6 louver blades installed: (a) d = 45 μm; (b) d = 80 μm; (c) d = 150 μm; (d) d = 400 μm; (e) d = 700 μm; (f) d = 1200 μm

In order to analyse the effects of particle size distribution on the concentration profile of pulverized coal, it was necessary to introduce a mono-dispersed flow of pulverized coal at the inlet of flow domain. In order to obtain flow distribution of poly-dispersed mixture to the burner feed ducts it was necessary to calculate the mass flow rate of particular fraction in burner ducts based on the related mass rate in the poly-dispersed mixture (fig. 2). The specified computation method is by all means the most accurate. In such approach, however, each fraction needs to be computed separately, therefore making the computation very time-consuming. The computation time may be reduced by analysing the flow of poly-dispersed mixture in the flow domain developed. The size of each particle can be generated stochastically, based on the particle size distribution presented in fig. 2. Thus, instead of computing every fraction separately, only one computation needs to be performed. The method based on a poly-dispersed flow computation is less accurate since the overall number of track particles in this situation is representative of not one, as in the case of mono-dispersed flow, but of all fractions considered. Still, if differences between the results of such computation and the results obtained by computing each fraction separately are found to be relatively small, the use of less time-consuming method would be appropriate. Comparison of the results obtained utilizing the two approaches considered is presented in fig. 6.

Analysis of the results obtained

In order to reach the main research objective, *i. e.* to increase the concentration of pulverized coal in the thermal plasma zones, it is necessary for the installed louver to induce appro-

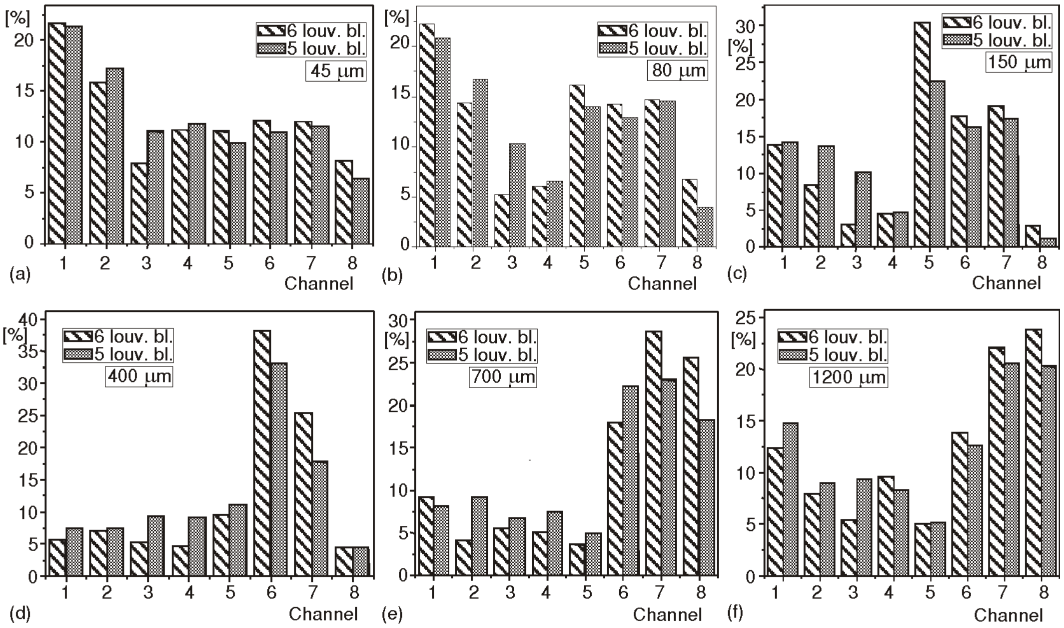

Figure 6. Normalized steady state distribution of total coal flow to the burner feed ducts (from the left side to the right) for 5 and 6 louver blades installed: (a) $d = 45\,\mu$m; (b) $d = 80\,\mu$m; (c) $d = 150\,\mu$m; (d) $d = 400\,\mu$m; (e) $d = 700\,\mu$m; (f) d = 1200 μm

priate phase separation. The said means that air must be allowed to pass between the louver blades. Numerical simulation with various louver types has shown that the key factor for efficient phase separation is a distance between two adjacent louver blades. If two adjacent blades are positioned too close to one another, the recirculation zone behind the blades could be large enough to completely prevent the flow. Analysis of the gas flow pattern shown in figs. 2b and 3 indicates that recirculation zones established were not wider than the louver blades. Therefore it is concluded that louver comprising not more than 6 blades does not prevent the air to flow into the plasma-free burner ducts, in that way enabling the required phase separation.

Qualitative analysis of the results presented in figs. 4 and 5 and quantitative analysis of those presented in fig. 7, where normalised steady-state distribution of pulverized coal to the burner ducts are shown for each of the six particle sizes selected, provided an insight into the relation between the coal concentration profile and the coal particle size. Figures 4 and 5 clearly indicate significant differences in distribution of coal particles of different sizes. The finest particles with the lowest inertia follow the streamlines of the continuous phase pretty accurately, diagonally passing between the louver blades. The said blades do not affect the particle motion. With an increase in the particle diameter the inertia of the particles increases and so does the horizontal distance the particle can travel after hitting the louver blade. In that way, particles are able to reach the burner ducts equipped with plasma generators more easily. In this manner the presence of louver increases the coal concentration in the considered ducts. The impact of the continuous phase on the largest particles is very small, so the particles either pass vertically between the louver blades or hit them and move horizontally, having enough inertia to reach the right side wall. The said behaviour is manifested by particle agglomeration behind the right side plate and considerable increase in the mass flow rate of pulverised coal in the ducts equipped with plasma generators. As seen in fig. 6, increased coal concentration in burner ducts equipped

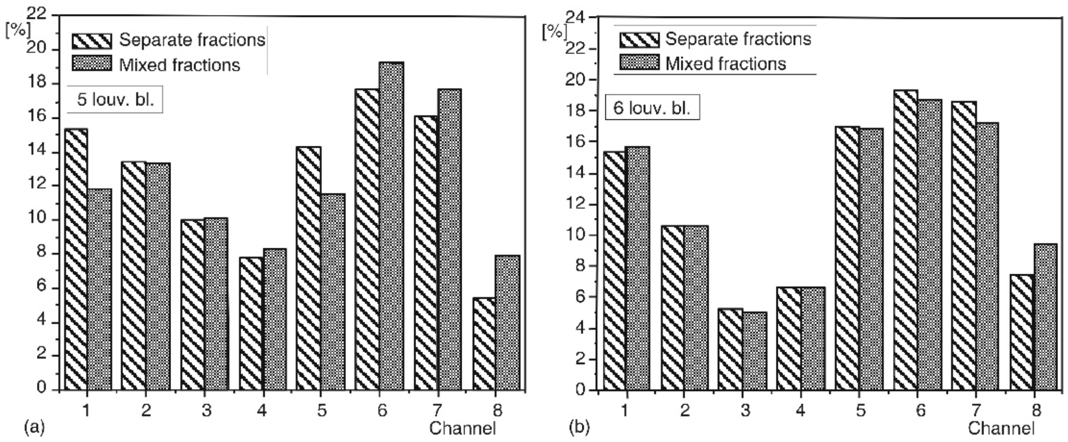

Figure 7. Normalized steady-state distribution of total coal flow to the burner feed ducts for: (a) 5 louver blades installed; (b) 6 louver blades installed

with plasma reactors is recorded for coal particles having a diameter of 150 μm and higher. Still, it is interesting to note that increase in particle diameter initially causes a drop in coal concentration in the last duct (closest to the right side wall), whereby the said parameter starts to increase only after the particle diameter reaches 700 μm. The explanation for such behaviour is found in a tendency of mid-size coal particles not to closely follow the gas streamlines, whereby the particles which hit the plate positioned to the right reflect toward the central part of the flow domain. At the same time, inertia of the mid-size particles hitting the louver blades is still not enough to enable them to pass across the entire width of the flow domain and reach the region to the right.

The discrete phase flow patterns shown in figs. 4 and 5 are quite similar. Differences in coal distribution in the two cases considered are relatively small (fig. 6). Coarser particles are more affected by an increase in the number of louver blades when compared to the fine particles. Mass flow rate in ducts equipped with plasma generators tends to increase with an increase in the number of blades.

Comparison of the results of coal distribution to the burner ducts obtained for simulated flow of poly-dispersed mixture in the flow domain *vs.* the simulated flow of mono-dispersed mixtures, fig. 7, indicate very good agreement between the results obtained with the two approaches used. In the case of six louver blades installed, the results obtained in the previously mentioned simulations were almost identical. In the case of five louver blades installed, the results obtained showed a certain, but quite small discrepancy. Based on the analysis performed it is generally concluded that less time consuming approach, *i. e.* poly-dispersed approach, can be used without significantly affecting the accuracy of the results obtained.

Conclusions

The paper addresses numerical analysis of the influence of particle fineness on the efficiency of louvers utilization for flow redirection, as well as phase separation in a duct located after the mill and before the burner feed ducts, with respect to the flow direction. The results obtained confirmed the general conclusion (based on the use of louvers in power plants worldwide) that louvers may be efficiently utilized as a tool for reaching the above mentioned goals which could not be achieved by the sole use of traditional classifiers. Installation of louvers ahead of the burner feed ducts results in high enough concentrations of pulverized coal in the ducts equipped with plasma generators for the successful fuel ignition to be accomplished. In this manner the principle objective of the research conducted has been successfully achieved. Still, combined and simultaneous use of both regulation systems, *i. e.* louvers and classifiers, should not be excluded. Care should be taken to insure that installation of louvers does not cause a reduction in mill capacities or self-ignition of pulverized coal.

Comparison of the results obtained by calculation of each particle fraction *vs*. the calculation of poly-dispersed mixture indicate small differences between the two methods employed, the difference being smaller than the computational error. Consequently, the use of poly-dispersed mixture provides a few times smaller computational time and is therefore recommended for future analysis.

With respect to the use of numerical simulation for analysing the influence of louver installation on the flow profile, it has been demonstrated that after the developed model has been verified by comparison with experimental data, numerical simulation may be utilized as a reliable and efficient flow analysis tool. Use of commercial software package FLUENT 6.3 was deemed appropriate for the task defined. Generation of numerical grid was quick and convergence was reached without any difficulties. Use of developed user-defined functions provided all necessary statistical flow field parameters to be obtained easily. In this manner, a necessity for expensive, numerous, long lasting and difficult experimental measurements is considerably reduced. Verified numerical model enables investigation of numerous flow patterns characterised by different flow parameters and geometry, providing considerable assistance in finding the optimal louver design. Moreover, demonstrated impact of louver geometry and position on flow characteristics, as well as provision of particular results which could hardly be intuitively expected, confirm that numerical simulation represents not only convenient, but necessary tool for finding the optimal problem solution.

The research results presented in the paper did not address the influence of louver installation on mill capacities and their proper operation. Measurements performed on mills following the louver installation indicated that mill operation has not been interrupted. However, there is a possibility that proper mill operation would be affected by installation of louvers characterized by different design parameters. Therefore, the investigation of louver utilization should be extended, further considering the coupled effects of particle fineness and louver parameters on the velocity and concentration fields. In addition, the numerical model employed should be extended so as to take into account mill characteristics and operation. In such analysis special attention should be devoted to additional pressure drop resulting from the louver installation.

Nomenclature

C_D – drag coefficient, [–]
d_p – particle diameter, [m]
$\vec{F}$ – additional forces, [N]
F_D – drag force, [N]
$\vec{g}$ – acceleration due to gravity, [ms^2]
Re – Reynolds number
$\vec{u}, \vec{u}_p$ – fluid and particle velocity, [ms^{-1}]

Greeck letters

ν – cinematic viscosity, [m^{2}s^{-1}]
μ – dynamic viscosity, [Pa s]
ρ, ρ_p – fluid and particle density [kgm^{-3}]
τ_p – particle relaxation time, [s]

References

[1] Živković, G., Nemoda, S., Radovanović, P., Numerical Analysis of the Influence of Stationary Louvers on the Coal Powder Distribution in the Channels for Coal-Air Mixture on the Boiler of TENT-B1, Internal Report (in Serbian), NIV-ITE 367, Vinča, Belgrade, 2008

[2] Sommerfeld, M., Živković, G., Recent Advances in the Numerical Simulation of Pneumatic Conveying through Pipe Systems, *Proceedings*, Computational Methods in Applied Science, Invited Lectures and Special Technological Sessions of the First European Computational Fluid Dynamics Conference, Brussels, 1992, pp. 201-212

[3] Živković, G., Mathematical Modelling of Two Phase Gas-Particle Flow in Horizontal Tubes and Channels, Ph. D. thesis (in Serbian), Faculty of Mechanical Engineering, University of Belgrade, 1996

[4] Pavlović, P., Stefanović P., *et al*., The Use of Plasma Torch for the Burning Process Stabilisation on the Boiler in the 210 MW Unit "Nikola Tesla" Power Plant, Technical Report (in Serbian) NIV-ITE-155, Vinča, Belgrade, 1999

[5] Stefanović, P., Pavlović, P., *et al*., Plasma Systems for the Ignition and Fire Support at the Boilers Fed by Coal Power Survey of the Existing Industrial Installations in the World (in Serbian), *Procesna tehnika*, *19* (2003), 1, pp. 138-142

[6] Morsi, S. A., Alexander, A. J., An Investigation of Particle Trajectories in Two Phase Flow Systems, *J. Fluid Mech., 55* (1972), 2, pp. 193-208

[7] Li, A., Ahmadi, G., Dispersion and Deposition of Spherical Particles from Point Sources in a Turbulent Channel Flow, *Aerosol Science and Technology, 16* (1992), 4, pp. 209-226

[8] Živković, G., Živković, N., Stefanović, P., Numerical Simulation of the Control of Coal Power Disribution in the System for the Fire Support in the Boiler, by Using Low Temperature Plasma (in Serbian), *Proceedings of Abstracts*, 11th Symposium of Thermal Science and Engineering in SCG, Zlatibor, 2003, p. 45

[9] Radovanović, P. *et al*., Thermal Characteristics of the Unit TENT-B1 620 MW before and after Refurbishment in 2005, with the Evaluation of Its Effects, Internal Report (in Serbian), NIV-ITE 310, Vinča, Belgrade, 2006

[10] Živković, G., Nemoda, S., Radovanović, P., Numerical Analysis of the Influence of Stationary Louvers on the Coal Powder Distribution in the Channels for Coal-Air Mixture on the Boiler of TENT-B1, Internal Report (in Serbian), NIV-ITE 361, Vinča, Belgrade, 2007

Author's affiliation:

G. Živković **(corresponding author)**
Laboratory for Thermal Engineering and Energy,
Vinča Institute of Nuclear Sciences
P. O. Box 522, Belgrade, Serbia
E-mail: gzivkovic@vinca.rs

S. Nemoda, P. Stefanović, P. Radovanović
Laboratory for Thermal Engineering and Energy,
Vinča Institute of Nuclear Sciences
Belgrade, Serbia

Paper submitted: January 15, 2009
Paper revised: January 26, 2009
Paper accepted: July 9, 2009