Turbulent Two-Phase Flow Modeling of Air-Coal Mixture Channels with Single Blade Turbulators

Rastko Jovanović
Research Scientist
Institute of Nuclear Sciences Vinča

Dejan Cvetinović
Research Scientist
Institute of Nuclear Sciences Vinča

Predrag Stefanović
Research Scientist
Institute of Nuclear Sciences Vinča

Bartosz Swiatkowski
Research Scientist
Warsaw University of Technology
Institute of Power Engineering

The subject of this work is turbulent two-phase flow through air-coal channel(s) of complex geometry. The aim of this work is numerical optimization of fluid flow and coal particle distribution in reconstructed air-coal mixture channels. The single blade turbulator has been used to increase turbulence in the vertical section of an air-coal mixture channel. Standard ***k-ω*** *turbulent model has been used for modeling turbulence. Lagrangian multiphase model has been used for discrete phase (coal particles) modeling. Although better particle distribution is reached using single blade turbulators, particle concentration in the evaluation section (where plasma generators will be built in) still remains anisotropic. Because uniform coal particle distribution is of great importance for the proper work of plasma generators, other solutions for achieving this goal will be the object of the future analysis.*

Keywords: *computational fluid dynamics (CFD), two-phase flows, Lagrangian multiphase model, single-blade turbulators.*

1. INTRODUCTION

Current fuel consumption for startup and combustion support in the Electric Power Utility of Serbia is about 100,000 tons per year, with a tendency to increase in the future due to decline in coal quality. Fifty percent of the fuel is consumed during startup, and the rest is used for combustion support when a unit is working with lower power production or with low quality coal. Existing oil burner system in Thermal power plant “Nikola Tesla” – A1, Obrenovac, Serbia, is planned to be supplemented with a new system for combustion support based on low temperature thermal plasma. Plasma system for fire stabilization of pulverized coal combustion on thermal power plant boilers has already been implemented throughout the world (USA, Russia, Australia, and China) [1].

In order to implement advanced plasma system for pulverized coal thermo chemical treatment on thermal power boilers, several technological and technical requirements are to be fulfilled. The most important is to obtain: desired air mixture flow in the air-coal mixture channel and desired pulverized coal particle distribution in the air-coal mixture channel operation [2].

Based on these requirements, the geometry of the existing rectangular channels has been changed. Lower four channels are divided in eight, with new geometry, in such a way that each existing channel with a rectangular cross-section is replaced with two new channels with a circular cross-section as it is shown in Figure 1.

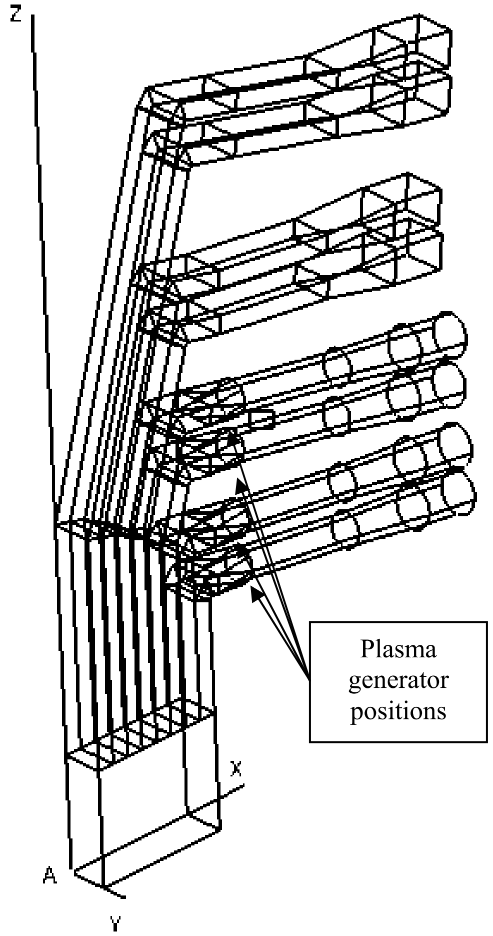

Figure 1. Basic Geometry with coordinate system

2. NUMERICAL MODEL

This model is analyzed in the global Cartesian coordinate system. The centre of the coordinate system is placed at the inlet (point A, Fig. 1), x-axis is oriented

---

Received: May 2008, Accepted: September 2008
Correspondence to: Rastko Jovanović
Institute of Nuclear Sciences Vinča,
Laboratory for Thermal Engineering and Energy,
Mike Alasa 12-14, 11001 Belgrade, Serbia
E-mail: rastko.jovanovic@ien.com.pl

in the direction of the horizontal parts of the channels. Only half of the complete air-coal mixture channels are used for numerical modeling, due to simplicity.

The main dimensions of the model used for the single phase flow modeling are: 6.67 m in x-direction, 0.6 m in y-direction and 10.61 m in z-direction. Flow through the channels of such complex geometry is expected to be three – dimensional, so 3D model with a large number of finite volumes is used for numerical modeling.

Hybrid mesh with the total number of 1,300,000 control volumes and with an average cell dimension of 4 cm is generated for the single-phase flow numerical simulation. Hybrid mesh with the total number of 350,000 control volumes and with an average cell dimension of 3 cm is generated for the two-phase flow numerical simulation. Used hybrid mesh consists of regular hexahedral (in regions with a rectangular cross-section) and tetrahedral cells (in regions with a round cross-section). At the section between the rectangular and the round cross-section (Fig. 1), the appropriate faces of transition cells are spited to make interface with triangular faces of tetrahedral. [3].

Special care is taken to precise geometrical interpretation of the evaluation section between the rectangular and the round cross-section in order to prevent possible numerical asymmetry in numerical results, not originating from the physical properties of the fluid flow.

The model is divided in to two computational domains for cases with a built blade turbulator. The turbulator is treated as a separate solid domain, extracted from the surrounding fluid domain. The fluid domain is divided into two separate sub domains. First (inner) sub domain encircles the turbulator. Mesh in this sub domain is changed in different computational cases, due to changes in turbulator geometry and position. Mesh in second (outer) sub domain remained the same for all computational cases with built turbulators, as shown in Figure 2. The described mesh treatment enables high accuracy in results comparison. An average cell size for solid domain is 0.8 cm. and for outer, fluid, sub domain 3 cm. Mesh for inner, fluid, sub domain is created using boundary layers. Boundary layers are used for solving fluid flow around the turbulator with better accuracy. Five boundary layers, near turbulator walls, are used with boundary layer width growth 1.3. Cells in first boundary layer have an average size of 0.8 cm. Cells at the walls of the inner fluid sub domain have an average size of 3 cm. Tetrahedral mesh is used in the inner fluid sub domain.

3. MATHEMATICAL MODELING

Standard k-ω has been used for turbulence modeling. This model is a two-equation turbulent model which includes two extra transport equations to represent the turbulent properties of the flow. This allows a two-equation model to account for history effects like convection and diffusion of turbulent energy.

The first transported variable is turbulent kinetic energy, k. The second transported variable in this case is the specific dissipation, $\omega \equiv \varepsilon / k$. It is the variable that determines the scale of the turbulence, whereas the first variable, k determines the energy in the turbulence [4]. Standard wall functions are used for the determination of flow field near the wall (except for turbulator walls, where boundary layers are used).

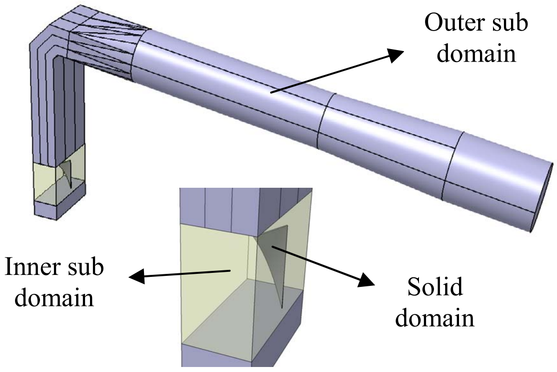

Figure 2. 3D model view with sub domains

SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) algorithm is for pressure-velocity coupling. The SIMPLE algorithm uses a relationship between velocity and pressure corrections to enforce mass conservation and obtain the pressure field. The approximation of the velocity field is obtained by solving momentum equation. The pressure gradient term is calculated using the pressure distribution from previous iteration or an initial guess. The pressure equation is formulated and solved in order to obtain a new pressure distribution. Velocities are corrected and a new set of conservative fluxes is calculated [5].

Disperse phase is modeled using the Lagrangian approach. In this approach a discrete phase particle trajectory (pulverized coal) is predicted by integrating the force balance on the particle. This force balance equates the particle inertia with the forces acting on the particle, and can be written (for the x direction in Cartesian coordinates) as:

$$\frac{\mathrm{d}u_{\mathrm{p}}}{\mathrm{d}t} = F_{\mathrm{D}}(u - u_{\mathrm{p}}) + \frac{g_x(\rho_{\mathrm{p}} - \rho)}{\rho_{\mathrm{p}}} . \qquad (1)$$

Here, u is the fluid phase velocity, u_{p} is the particle velocity, μ is the molecular viscosity of the fluid, ρ is the fluid density, ρ_{p} is the density of the particle, and d_{p} is the particle diameter. F_{D} $(u - u_{\mathrm{p}})$ is the drag force per unit particle mass. Two-way coupling between disperse (pulverized coal particles) and fluid phase is included. This means that momentum transfer from the continuous phase to the discrete phase is computed by examining the change in the momentum of a particle as it passes through each control volume. It has been assumed that this momentum exchange happens every 30 fluid flow iterations.

The 2500 computational particles are used to represent physical particles in this paper. It is assumed then that a parcel of particles moves through the field with the same velocity and temperature as a single particle (physical particle). Size distribution effects have to be included specifying parcels with a specific particle size. The Rosin-Rammler diameter distribution is used in this paper (Table 1). Coal particles are injected from the inlet surface in the fluid domain with the same velocity magnitude. Additional data for particle injection are given in Table 2.

Table 1. Rosin-Rammler diameter distribution

Minimal diameter [m]	0.00018
Maximal diameter [m]	0.002
Mean diameter [m]	0.0004
Spread parameter	3.5
Number of diameters	20

Table 2. Particle injection properties

Velocity component in *x*-direction [m/s]	0
Velocity component In *y*-direction [m/s]	0
Velocity component In *z*-direction [m/s]	18
Total flow rate [kg/s]	1.008

The trajectory is obtained from:

$$\frac{\mathrm{d}x_{\mathrm{P}}}{\mathrm{d}t} = v\,. \qquad (2)$$

The following turbulent model characteristics are used for all cases:

- standard pressure discretization;
- first-order upwind momentum, *k*, *ω* discretization;
- for both cases all flow properties are defined with residuals less than 1E-5.
- In the used mathematical model the following assumptions are introduced:
  - incompressible flow (ρ = const);
  - isentropic flow ($\delta Q = 0$);
  - gravity acceleration is set at -9.81 m/s along *z*-axis.

Boundary conditions are set to:

- uniform velocity field at inlet cross-section with the velocity magnitude of 18 m/s derived perpendicularly to the inlet cross-section;
- pressure outlets with gauge pressure value 0 Pa;
- no slip is considered.

4. RESULTS AND DISCUSSION

One-phase flow through all eight air-coal mixture channels, with air as working fluid, has been calculated in the first case. Velocity and static pressure values are shown in Figures 3 and 4.

It can be seen that obtained values for static pressure are between – 400 Pa and 200 Pa. Figures 8 and 10. show negative values of static pressure, which derive from boundary conditions at channel outlets. Results pointed out that high static pressure drop exist in the vertical section of first two channels due to sudden change in flow direction (more than 90 degrees). This static pressure drop in vertical sections of other channels is not that high, because in vertical sections of these channels, before sudden change at crossing from vertical in horizontal section, exists small change of direction (less than 10 degrees), as shown in Figure 4.

Values of velocity magnitude follow values of static pressure as can be seen in Figure 7. Values of velocity magnitude change between 0 m/s and 26 m/s, as shown in Figure 3. and its values follow static pressure distribution, as shown in Figure 4. High velocity magnitude values, between 16 m/s and 26 m/s are distributed in upper channel cross-sections. Lower cross-sections have velocity values up to 10 m/s, and central cross-sections have velocity values approximately between 10 m/s and 16 m/s.

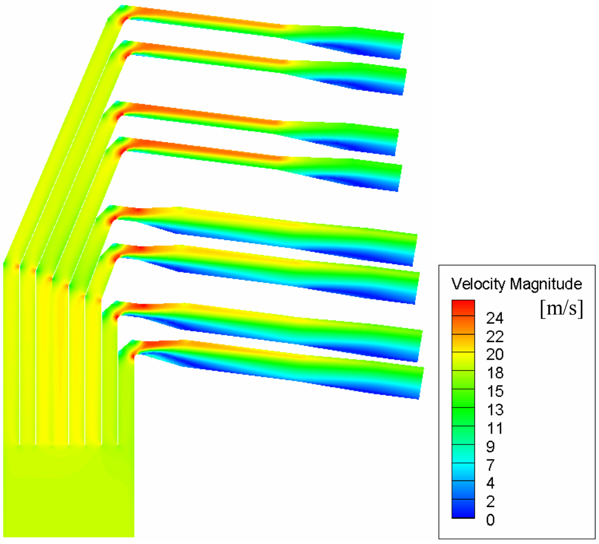

Figure 3. Velocity magnitude values at central cross-section, plane *y* = 0

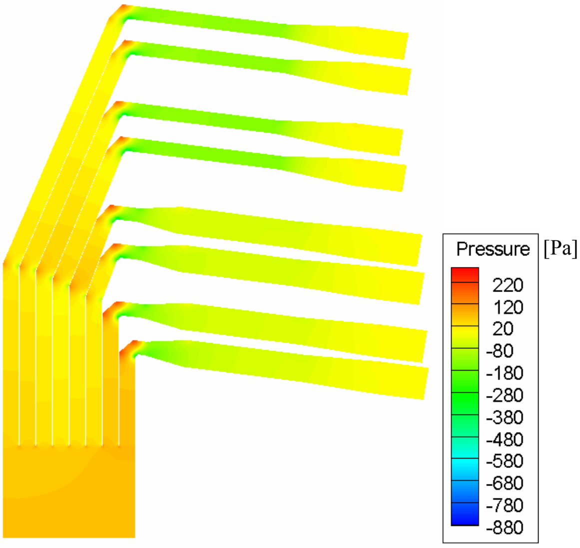

Figure 4. Static pressure distribution at central cross-section, plane *y* = 0

It is expected that high values of static pressure values in vertical sections of two lowest channels will cause higher coal particles concentration in upper sections at the beginning of horizontal parts of channels. Plasma generators will be built in these sections, so uniform coal particle distribution in these sections is of great importance.

The lowest channel has been chosen for two-phase flow analysis in order to investigate coal particle distribution.

Pressure distribution at central cross-section is shown in Figure 5. Velocity magnitude at central cross-section is shown in Figure 6. Concentration of coal particles (in percents) is shown in Figure 7.

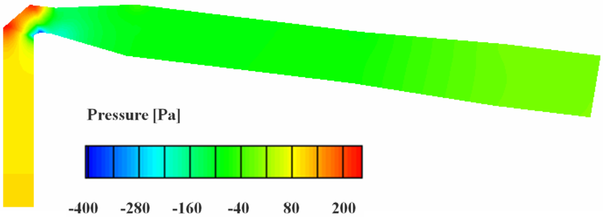

Figure 5. Pressure distribution at central cross-section, plane $y = 0$

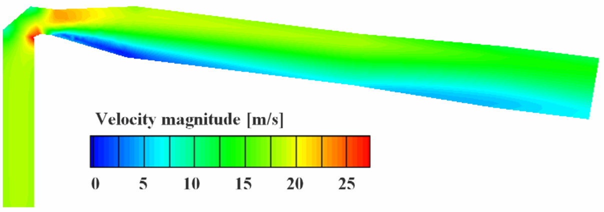

Figure 6. Velocity magnitude at central cross-section, plane $y = 0$

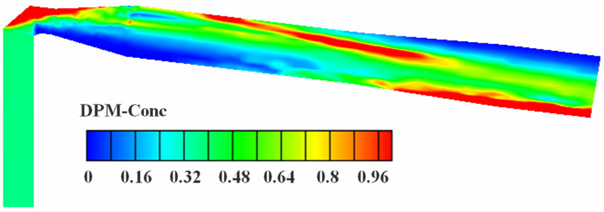

Figure 7. Discreet phase (coal particles) concentration at central cross-section, plane $y = 0$

Velocity and pressure values are similar to those in one-phase flow case. Coal particles are uniformly distributed in vertical part of channel, with concentration around 0.5 %. The zone with highly anisotropic coal particle distribution is located at evaluation section between vertical and horizontal part of channel, as it is expected (Fig. 7). This non-uniform distribution is the result of strong particle inertia. Coal particles have better distribution in the middle part of horizontal section. Particles are mainly concentrated in the lower parts of the channel in the last section of horizontal part of channel (as it can be seen in Figure 7). The reasons for this are gravity acceleration and collision between coal particles and channel walls. Coal particles reflect from the wall after their collisions with channel walls.

Uniform distribution of coal particles in evaluation section between horizontal and vertical part of channel is of great importance, because plasma generators will be built in this section. Single blade turbulators are considered for implementation in vertical section of channels (Fig. 2). The purpose of these turbulators is to obtain more uniform coal particle distribution. Turbulator should give swirl to air-coal flow and in that way increase mixing between air and coal phase. Particle paths will be also increased in this way, which will give additional time for plasma generators to gasify pulverized coal. Turbulator geometry is given in Figure 8.

Two different single blade turbulator geometries were considered. The following values were the same for both cases: turbulator length $l = 300$ mm, turbulator thickness $\delta = 5$ mm and inflection angle $\alpha = 45$ deg. Turbulator height was $h = 63$ mm in normal projection (equaled to 1/4 of channel height) for the first case, and $h = 126$ mm in normal projection (equaled to 1/2 of channel height) for the second case (Fig. 10). Turbulator length was positioned parallel to velocity vectors in inlet for these two cases.

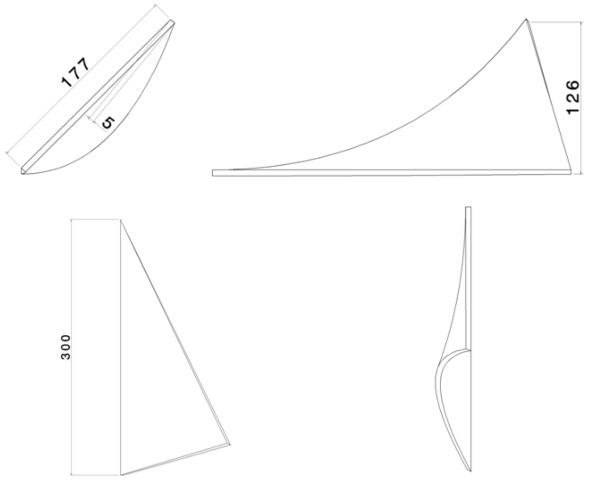

Figure 8. Single blade turbulator geometry

Finally, turbulators were rotated for 12 deg. around x-axis, in positive mathematical direction, and simulation was repeated for both turbulator heights.

Velocity contours for the first two cases (turbulator length parallel to velocity vector at inlet cross-section) are shown in Figure 9. Velocity contours for the third and fourth case (turbulator rotated around x-axis for 12 deg. in positive mathematical direction) are shown in Figure 10. Turbulence intensity is defined as

$$I \equiv \frac{u'}{U} \cdot 100\,\%\,,$$

where u' is the root-mean-square of the turbulent velocity fluctuations and U is the mean velocity (Reynolds averaged). Turbulence intensity, for all five investigated cases, is given in Figure 11. Total discrete phase concentration is defined as a mass of discrete phase per fluid volume, kg/m3. Normalized value of total discrete phase concentration has been used for results presentation. Normalized discrete phase concentration is equal to total discrete phase concentration divided with its maximum value (maximum value is equal for all cases). Thus, relative discrete phase concentration can obtain values between 0 and 1. Values of discrete phase concentration, for all five investigated cases, is given in Figure 12.

Values of velocity magnitude are similar for all five cases. Velocity values are around 18 m/s in vertical part of the channel, and velocity distribution is almost uniform in this part of the channel. There is slightly an increase of velocity magnitude around turbulator in cases in which turbulators are built in. Velocity increases up to 25 m/s near right channel wall in the section near sudden change of flow direction (section between vertical and horizontal part of channel). Velocity decreases to 12 m/s near the left channel wall in the same position. Velocity distribution in the evaluation section, the channel section between rectangular and round cross-section, differs between case without turbulator and cases with built turbulators.

Higher velocity values, around 22 m/s, are concentrated in the upper parts of evaluation section, while lower velocity values, around 5 m/s are distributed near lower parts of evaluation section. This zone with anisotropic velocity distribution, in case without turbulator (Fig. 6), is mainly located in the first half of evaluation section, and in cases with tabulators (Figs 9 and 10) it spans through the whole evaluation section. Velocity field becomes more uniform, approaching channel outlet.

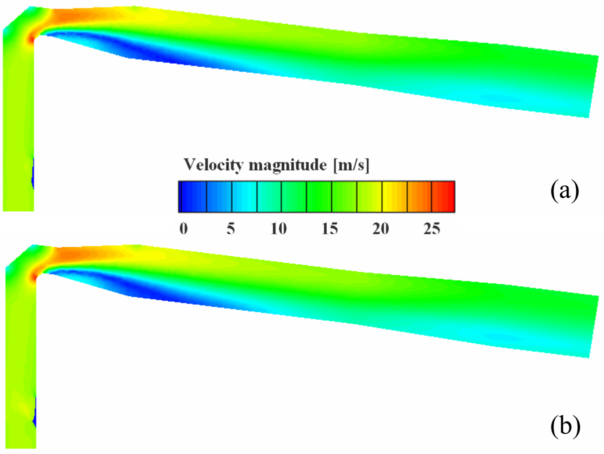

Figure 9. Velocity magnitude at central cross-section, plane *y* = 0. Turbulator length set parallel to velocity vector at inlet cross-section. (a) turbulator height *h* = 63 mm; (b) turbulator height *h* = 126 mm

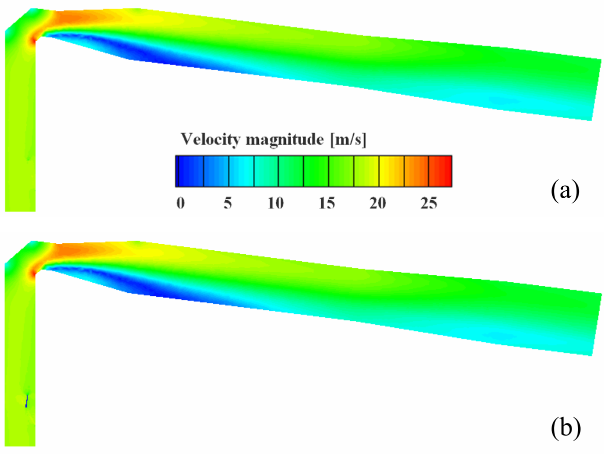

Figure 10. Velocity magnitude at central cross-section, plane *y* = 0. Turbulator length rotated around *x*-axis for 12 deg. in positive mathematical direction. (a) turbulator height *h* = 63 mm; (b) turbulator height *h* = 126 mm

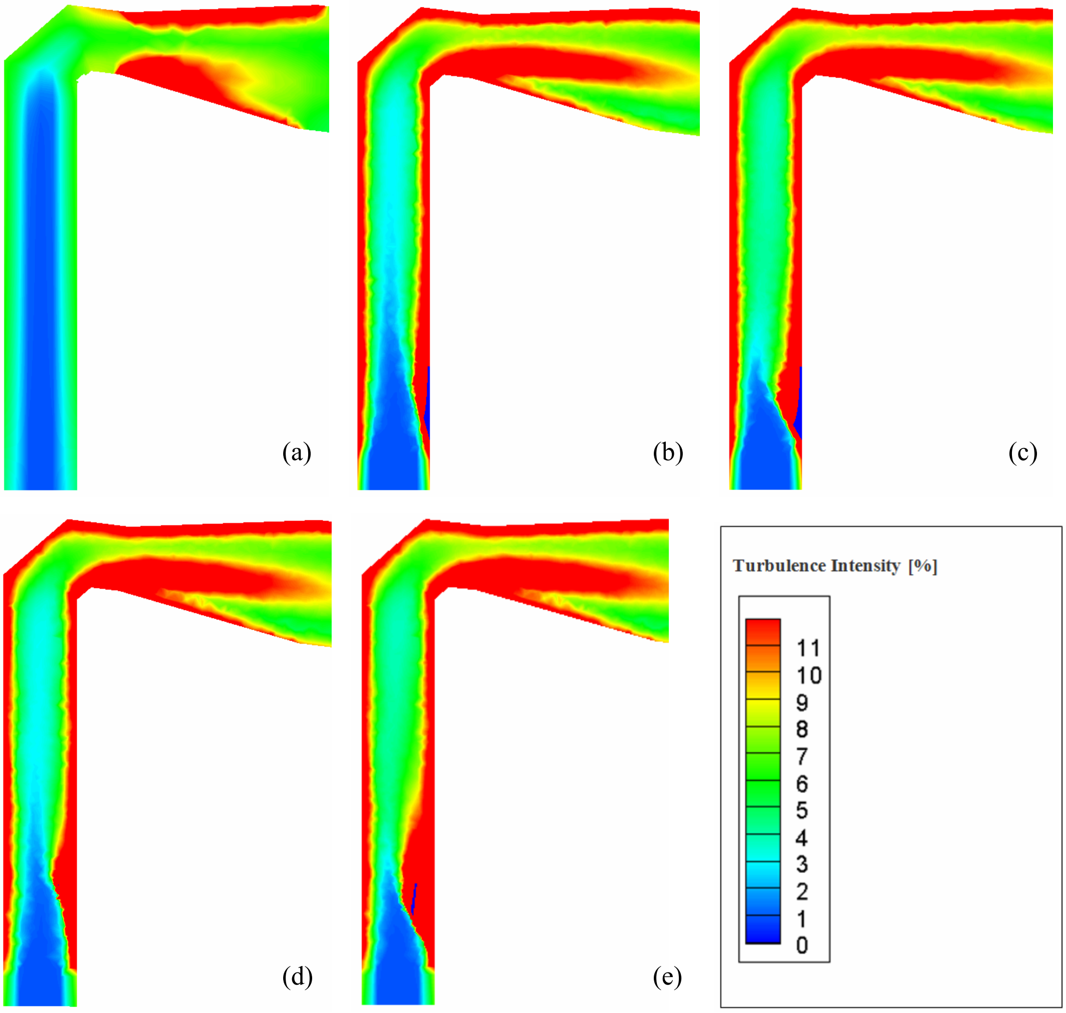

Figure 11. Turbulence Intensity at central cross-section. (a) case without turbulator; (b) case with turbulator height *h* = 63 mm, turbulator positioned parallel to inlet velocity vector; (c) case with turbulator height *h* = 126 mm, turbulator positioned parallel to inlet velocity vector; (d) case with turbulator height *h* = 63 mm, turbulator rotated for 12 deg. around *x*-axis in positive mathematical direction; (e) case with turbulator height *h* = 126 mm, turbulator rotated for 12 deg. around *x*-axis in positive mathematical direction

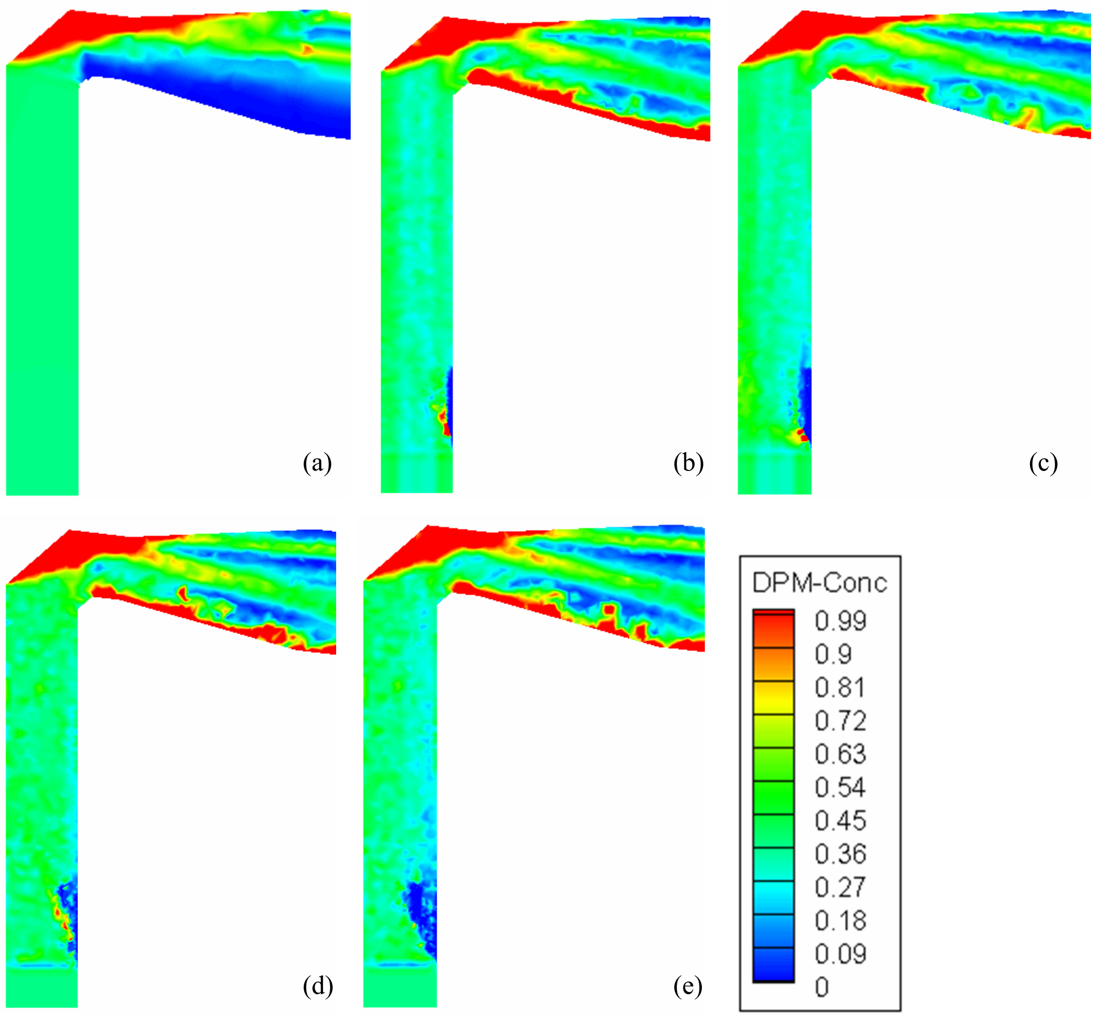

Figure 12. Discrete-phase concentration at central cross-section. (a) case without turbulator; (b) case with turbulator height *h* = 63 mm, turbulator positioned parallel to inlet velocity vector; (c) case with turbulator height *h* = 126 mm, turbulator positioned parallel to inlet velocity vector; (d) case with turbulator height *h* = 63 mm, turbulator rotated for 12 deg. around *x*-axis in positive mathematical direction; (e) case with turbulator height *h* = 126 mm, turbulator rotated for 12 deg. around *x*-axis in positive mathematical direction

A clearer picture of turbulator influence on flow field can be obtained from turbulence intensity values.

The case without turbulator is given in Figure 11a. The zone with very low turbulence intensity, around 1.2 %, is located in the middle section of vertical part of the channel. The zones with middle values of turbulence intensity, around 6 %, in vertical part of the channel, are located near left and right channel wall. Turbulence intensity has the value around 5 % in the middle of evaluation section of channel, and high values, around 11 %, are located near the upper and lower walls of evaluation section of the channel.

The case with turbulator with lower height h = 63 mm, set up parallel to velocity vectors at inlet is given in Figure 11b. The zone with low turbulence intensity, around 1.2 %, is wider in channel inlet comparing with previous case, but farther in positive z direction, turbulence intensity increases, and obtains values between 3 % and 6 % in the middle of vertical part of the channel. Turbulence intensity has higher values, around 11 %, near left and right channel walls in vertical part of the channel, comparing with case without turbulator. Turbulence intensity distribution and magnitude is also different in evaluation section, comparing with case without turbulator. Highest values of turbulence intensity, 11 %, are located in the middle part of evaluation section, and near the upper and lower (in very thin layers) walls. Zones with middle values of turbulence intensity, round 6 %, in evaluation channel section, are located between zones with high turbulence intensity.

The case with turbulator with higher height h = 126 mm, set up parallel to velocity vectors at inlet is given in Figure 11c. Turbulence intensity distribution is similar to the case with turbulator with lower height, especially in evaluation section, in which turbulence intensity magnitude and distribution are almost the same for these two cases. Turbulence intensity increases in vertical part of the channel, comparing with case with lower turbulator height. The zone with low turbulence intensity is decreased.

Cases with rotated turbulators are given in Figures 11d and e. Turbulence intensity increases (for the same turbulator height) comparing with cases in which turbulators were set parallel to velocity vectors at channel inlet. This increase in turbulence intensity is especially intensive in the case of turbulator with higher height. The zone with high turbulence intensity, around 11 %, near turbulator is much wider than in the case with turbulator of the same height set parallel to velocity vectors at channel inlet.

Discrete phase concentration contours for all cases are given in Figure 12.

The case without turbulator, Fig. 12a, shows very anisotropic particle concentration in evaluation section of the channel. Coal particles are mainly located near the upper wall in first half of evaluation section, and discrete phase concentration has very high values, almost 1. Middle values of discrete phase concentration, around 0.5, are located in the upper and middle part in the second half of evaluation section. Lowest values of discrete phase concentration, around 0.08, are mainly located in the lower part of evaluation section.

Discrete phase concentration for all cases with turbulator is very similar. Particle distribution in evaluation section of the channel stays anisotropic, but this anisotropy is not as intensive as in the case without turbulator. The zone with very high discrete phase concentration, up to 1, still exists near the upper wall in the first half of evaluation section, but its dimensions are lower. Another zone with high discrete phase concentration, for cases with turbulator, is developed near the lower wall of evaluation section, but it is very thin. Values of discrete phase concentration in biggest part of evaluation section are between 0.2 and 0.5, which shows that increase of turbulence due to turbulators helps in achieving better particle distribution.

5. CONCLUSIONS

The aim of this work was numerical optimization of fluid flow and coal particle distribution in reconstructed air-coal mixture channels in thermal power plant “Nikola Tesla” – A1, Obrenovac, Serbia, using single blade turbulators to increase turbulence in vertical section of air-coal mixture channel. Standard k-ω turbulent model was used for modeling turbulence. Lagrangian multiphase model was used for modeling coal particle distribution.

Five different cases have been calculated in total. One case with two-phase flow in air-coal channel without turbulator, and two different turbulator geometries in two different positions have been analyzed. The analysis of the results shows that there is no significant change in velocity magnitude when turbulators are used. However, obtained results show that turbulators have a great influence in the increase of turbulence in vertical part of air-coal mixture channel. Turbulence increases with increase of turbulator height and increase of rotation angle. The highest level of turbulence increase has been reached when turbulator with higher height was rotated around x-axes, for 12 deg., in positive mathematical direction.

More uniform coal particle distribution was achieved using single blade turbulators. Results show that there is no significant difference in coal particle distribution between all four cases in which different turbulator geometry and position was used. Upon these conclusions, technologically simplest solution, turbulator with a low height, can be suggested.

Although better particle distribution has been reached using single blade turbulators, particle concentration in evaluation section (where plasma generators will be built in) still remains anisotropic. Because uniform coal particle distribution is of great importance for proper work of plasma generators, other solutions for achieving this goal will be the object of future analysis.

One of new approaches can be changing channel geometry. Geometry will be changed in the section in which vertical part of channel evaluates in horizontal. Channel cross-section area should be increased, and due to this, it is expected that velocity magnitude will decrease. Lower velocity magnitude should give longer time for plasma-generators to completely gasify pulverized coal.

Better results can be achieved with the use of single blade turbulators in combination with above mentioned geometry changes.

REFERENCES

[1] Stefanovic, P., Pavlovic, P., Cvetinovic, D., Zivkovic, N., Pavlovic, Z. and Markovic, Z.: Plasma systems for start-up and combustion support in pulverized coal fired boilers – review of existing world industrial facilities, Procesna tehnika, Vol. 19, No. 1, pp. 138-142, 2003, (in Serbian).

[2] Živković, N., Živković, G. and Stefanović, P.: Numerical simulation of coal powder distribution along the air-coal mixture channel influenced by various shutter angles in air − mixture channel on thermal power plant 210 MW TENT − A, in: *Proceedings of the International Scientific Conference MECHANICS 2006*, 2006, Rzeszów, Poland, pp. 261-268.

[3] Feldman, B.E., O'Brien, J.F. and Klingner, B.M.: Animating gases with hybrid meshes, in: *Proceedings of SIGGRAPH2005 the 32nd International Conference on Computer Graphics and Interactive Techniques*, 31.07-04.08.2005, Los Angeles, pp. 904-909.

[4] Davidson, L.: *An Introduction to Turbulence Models*, Publication 97/2, Department of Thermo and Fluid Dynamics, Chalmers University of Technlology, Göteborg, 2003.

[5] FLUENT 6 User’s Guide, Fluent Inc., 2006.

[6] Loth, E., Tryggvason, G., Tsuji, Y., Elghobashi, S.E., Crowe, C.T., Berlemont, A., Reeks, M., Simonin, O., Frank, Th., Onishi, Y. and van Wachem, B.: Modeling, in: Crowe, C.T. (Ed.): *Multiphase Flow Handbook*, CRC Press, Boca Raton, pp. 13-60-13-83, 2006.

МОДЕЛИРАЊЕ ТУРБУЛЕНТНОГ ДВОФАЗНОГ ТОКА АЕРО-СМЕШЕ СПРАШЕНОГ УГЉА У ГОРИОНИЧКИМ КАНАЛИМА СА ЈЕДНОСТЕПЕНИМ ТУРБУЛАТОРИМА

Растко Јовановић, Дејан Цветиновић, Предраг Стефановић, *Bartosz Swiatkowski*

Предмет овог рада је турбулентно двофазно струјање кроз горионичке канале аеро-смеше спрашеног угља комплексне геометрије. Циљ овог рада је нумеричка оптимизација струјног тока и расподеле честица спрашеног угља у реконструисаним горионичким каналима. За повећање турбуленције, у вертикалном делу горионичког канала аеро смеше уграђен је једностепени турбулатор. За моделирање турбуленције коришћен је стандардни **k-ω** турбулентни модел. Лагранжеов приступ је коришћен за моделирање секундарне фазе (честица спрашеног угља). Иако је употребом једностепених турбулатора постигнута боља расподела честица спрашеног угља, концентрација честица у прелазном делу (у коме ће бити уграђени плазма генератори) остаје неравномерна. Како је равномерна расподела честица спрашеног угља од есенцијалног значаја за правилан рад плазма генератора, друга решења за постизање равномерне расподеле честица ће бити предмет будуће анализе.