TECHNIUM
Technium Vol. 2, Issue 7 pp.94-106 (2020)
ISSN: 2668-778X
www.techniumscience.com

Numerical modeling of multiphase flow inside aero-mixture channel and low emission burner of boiler OB-650

Amel Mešić[1], Izudin Delić[1], Nedim Ganibegović[1]

[1]Faculty of Mechanical Engineering, University of Tuzla, Urfeta Vejzagića 4, 75000 Tuzla, Bosnia and Herzegovina
izudin.delic@untz.ba

Abstract. Determination of multiphase flow inside PC boiler plant is of particular importance for the process control of the boiler and its efficient operation. Nowadays numerical modeling is used as an advanced tool in improvement of this or similar process. Separation of coal particle in aero-mixture channel, after pulverization, represent an important process which has a big effect on boiler efficiency, and its determination represents an important step. In this paper, numerical modeling of multiphase flow inside aero-mixture channel and low emission burner of boiler OB-650 are exposed in several steps from 3D modeling, discretization of fluid domain, setting the physical and mathematical model to validation of same model. Main goals of the same process is to obtain valid numerical model of observed problematic, that will give us data about model parameters that can be used for modeling of the same process with different inlet boundary conditions, and also to obtain appropriate specific process parameters that can be used for rising of level of efficiency and utility of boiler plant in some steady operating modes.

Keywords. Numerical modeling, Coal boiler, Multiphase flow, Aero-mixture.

1. Introduction

Multiphase flow can be determinate inside the aero-mixture channel and low emission burner by using software's for numerical simulations. Many researchers have shown an interest in CFD (Computational Fluid Dynamics) simulation of such or similar problems. Of course, simulation of such complex problems wouldn't be possible without extensive development of commercial CFD codes and PC computers. Nowadays commercial CFD codes have capability to solve very fast and very complex equations for the conservation of mass, momentum, and energy, and to predict temperature, velocity, pressure and other required profiles.

It is important to emphasize that in previous reconstructions and modernizations of observed channel and low emission burner of boiler OB-650, this method of determination wasn't used, even thought, this was proven to be significantly useful method in practice. Below, some of the conducted research, that also provide the basic of this paper, have been mentioned. Stupar *et al. in* their paper [1] has studied numerical analysis of multiphase flow inside the elbow duct behind the mills aero-mixture separator in Thermal Power Plant "Kostolac A". In the same paper, optimal tilt of the elbow duct was determinated, and the reconstruction of the same channels was carried on, on the basic of the given results. Different elbow duct tilt has enabled better working conditions when mills are working on lower load, and deposition of the pulverized coal was prevented. Kozić *et al.* [2] investigate application of Euler-Euler approach for numerical modeling of multiphase flow inside the ventilation mill and aero-mixture channels with centrifugal separator of the Thermal Power Plant "Kostolac B". The same paper has

Technium Vol. 2, Issue 7 pp.94-106 (2020)
ISSN: 2668-778X
www.techniumscience.com

shown that this approach of numerical modeling gives us a satisfying results with smaller deviations. Kozić *et al.* [3] analyses the implementation of Eulerian and Lagrangian approach for numerical modeling of multiphase flow inside the ventilation mills. The same analysis has shown better superposition of numerical results with experimental measurements when Lagrangian approach is used. Milanović, in his PhD dissertation, [4] investigates turbulent multiphase flow in the straight channels of non-circular section. Živković *et al.* in their paper [5] and S. Atas *et al.* in [6] investigate the impact of dumpers and granulation of pulverized coal on the separation and the dispersion of coal particles over the burners in power plant, by CFD analysis. They concluded that the coal particle diameter represents main parameter that determinate the dispersion of coal flow. Kozić *et al.* presented the research [7] that deals with numerical investigation of impact of centrifugal separator on pulverized coal particle dispersion. Živković *et al*. conducted the research [8] that deals with numerical simulation of multiphase flow of aero-mixture channels of Thermal Power Plant “Nikola Tesla”, block A1. Babić *et al*. [9] were considering numerical simulation of multiphase flow across the mills separator type VML.210.50, with goal to achieve the best coal dispersion and separation. CFD modeling is already proven toll in this field (see, as example, Ferrin and Saavedra [10], Dodds *et al.* [11], Vuthaluru *et al*. [12] or Arakaki et al. [13]), and it is extensively used for optimization of PC burner design. Other similar types of scientific papers were proposed in references [14-20]. From previous papers, it can easily be seen how CFD is actually very useful in determination and prediction of various variables of multiphase flow, which cannot be determinated and predicted with classical methods. Of course, applicability and proper prediction of various process variables were the main goals of the numerical modeling, and the main goal of this paper. Gained experience with this method is applicable on the other boiler plants that have same or similar problems.

2. Numerical model

2.1. Geometry modelling and discretization of the fluid domain

Modeling of geometry for numerical analysis represents the first and probably, one of the easiest steps in the whole process of numerical modeling. Even though this step of modeling and discretization represents the easiest step, it isn’t nothing less important. Quality analysis of observed problematics can give us some conclusions about how important are some construction elements, and based on this information’s it is possible to generate optimal geometric construction for further discretization and numerical analysis, which will give us quality and valid results. The 3D model of aero-mixture channel and low emission burner has been made due to the technical documentation of boiler OB-650. It is important to emphasize that even though software for numerical analysis gives us opportunity for 3D modeling of analysied geometries. Detailed view of the modeled construction is shown in Fig. 1. In the same figure we can see the main elements of observed construction, such as manual regulation dampers, tubes for primary air, aero-mixture and air jets etc.

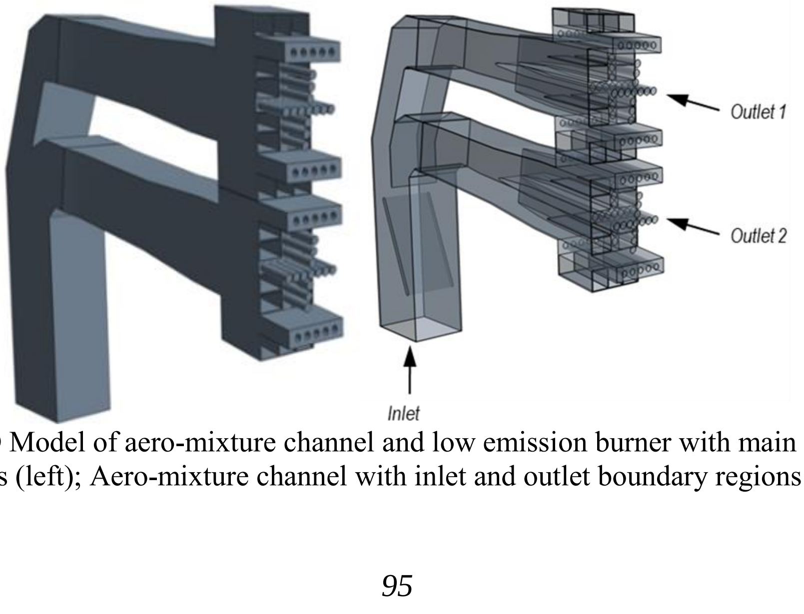

Figure 1. 3D Model of aero-mixture channel and low emission burner with main construction elements (left); Aero-mixture channel with inlet and outlet boundary regions (right).

As it is shown in Figure 1 (left), the construction is relatively good in the view of the complicity, but the global dimension of the whole construction is big. Global dimension and the nature of analyzed flow complicate the realization of the same analysis because it demands lager PC performances. Before spatial discretization of analyzed domain it is important to define physical boundaries of the system. Due to the previous in the Figure 1 (right) are shown inlet and outlet boundaries that were used during the analysis.

Discretization of observed construction can be done in different ways, with different basic volumetric elements (tetrahedral, hexahedral and polyhedral on Figure 2.). It is important to investigate which mesh type will give us best quality results, because quality of obtained results is the direct function of used mesh.

According to the previous research and examinations [1-20], preliminary analysis of discretization models has been conducted. The main parameters, that have been considered, were time of mesh generation, accuracy of the results, convergence, PC reassures and visual quality of obtained results. Of course, optimal solution can be obtained taking into account previous criteria, and criteria defined in the user guide of the STAR CCM+ [21].

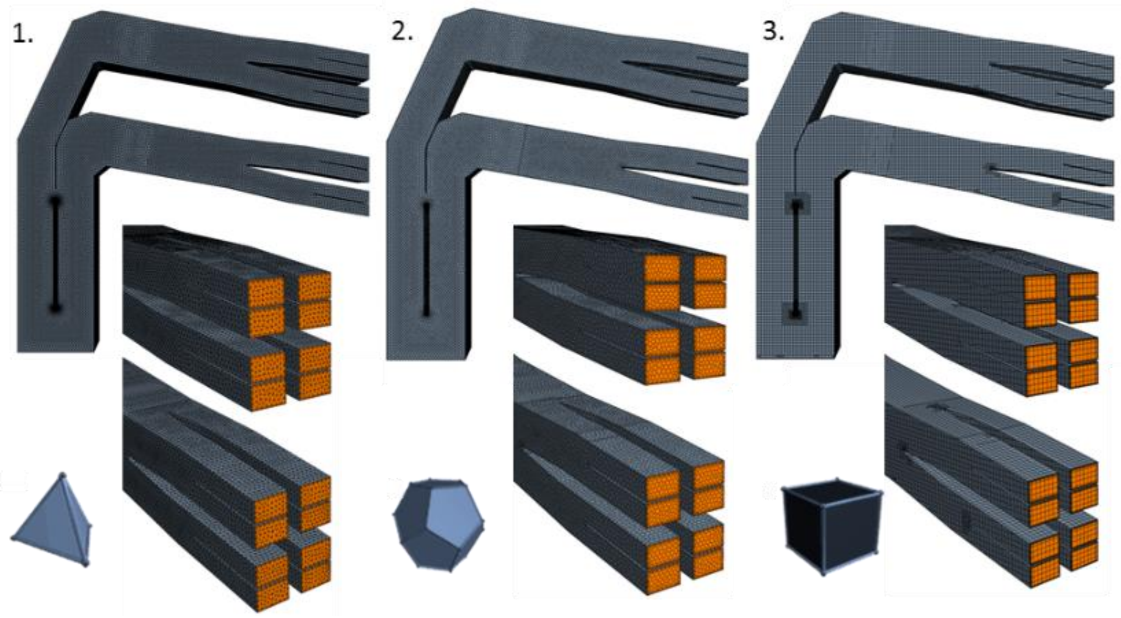

Figure 2. Fluid domain discretized with different approaches (1. Tetrahedral; 2.Hexahedar; 3.Polihedar).

Research that has been conducted on the analyzed construction has shown that big problem of convergence has polyhedral mesh. Of course, that was the main reason to eliminate this type of discretization, even though this type of discretization gives us best results in visually aspect. Better convergence has been achieved when tetrahedral and hexahedral mesh is used. On the Fig. 2 has shown fluid domain discretized with different approaches, where point 1 represent fluid domain discretized with tetrahedral mesh point 2 represent fluid domain discretized with polyhedral mesh, and point 3 fluid domain with hexahedral mesh.

Certainly, discretized domains shown in Fig. 2. were constructed with respect to the all demanded requires [21], that have to be fulfilled to analyze this problematics. Even though, the examinations of influence of the quality and type of the mesh on obtained results, weren't the scope of this paper, selection of appropriate type of mesh were unavoidable step, due to the fact that time of convergence depends on number of cells. Conducted preliminary analysis has shown that good convergence can be achieved both in a case of tetrahedral and the case of hexahedral mesh, but quality of the obtained visual results is much better in case of tetrahedral mess, so tetrahedral mesh was chosen for this analyses.

2.2. *Formulation of physical and mathematical model*

Optimal discretization and quality analysis of physical process of the multiphase flow are major front steps before determination of physical and mathematical model. Of course, mathematical model represents mathematical formulation of the physical model, which is, on the other side based on the series of the assumptions and hypothesis. During the whole process, it is necessary to take into account balance between complexity and excessive simplicity of both models. Analyzed problematic is further complicated due to the fact that instead of definition of one physical and mathematical model, it is necessary to define two models for two flows and their interaction, in this case.

Because, in the analyzed case, we are taking into account multiphase flow with lower concentration of disperse phase, and due to the criteria for selection of multiphase model and the previous theoretical researchs [1-21], Euler-Lagrangian approach is used for modeling. Where Eulerian approach is used for modeling primary flow, and Lagrangian approach is used for modeling dispersed flow.

In accordance with previous research, and as well with experimental parameters for better and complete defining of physical models of the primary and dispersed flow, the following assumptions have been adopted:

- For description of flow, continuum concept has been adopted;
- One component gas is taken into account;
- Gas flow is stationary, tridimensional, isotherm, incompressible, chemically inert and turbulent.

For defining physical model of secondary phase the following assumptions have been adopted:

- Particles are made from one material;
- Particles dimensions are approximated with spherical shape, and their dimensions are different;
- Mass of the particles remains constant during the flow through the aero-mixture channel;
- Particles have constant temperature and density;
- Two way coupling of phase has been taken into account;
- Particles lose one part of their kinetic energy during the collision with the walls ;
- Particles are moving stochastically.

Mathematical model of primary (gas) and dispersed (pulverized coal) flow will be formulated for conditions listed in the physical model, and it will be considered fully develop turbulent flow inside aero-mixture channel.

The main equations of mass, momentum and energy conservation for primary phase is identical with averaged general equation of conservation for single-phase fluid with addition of interphase member:

$$\frac{\partial}{\partial t}(\rho\Phi) + U_j\frac{\partial}{\partial x_j}(\rho\Phi) - \frac{\partial}{\partial x_j}\left(\Gamma_\Phi\frac{\partial\Phi}{\partial x_j}\right) = S_\Phi + S_\Phi^{IF} \quad (1)$$

where Φ is universal parameter of gas phase, ρ is a density of gas phase, U_j components of averaged velocities of gas phase, Γ_Φ transport diffusion coefficient of parameter φ, $\overline{S_\Phi}$ positive or negative source of parameter Φ, S_Φ^{IF} interphase member that describes interaction between the phases.

So basic equations of mass, momentum and energy equations respectively are:

$$\frac{\partial U_j}{\partial x_j}=0 \quad (2)$$

$$U_j\frac{\partial U_i}{\partial x_j} - \frac{\partial}{\partial x_j}\left(\nu\frac{\partial U_i}{\partial x_j}\right) = -\frac{1}{\rho}\frac{\partial P}{\partial x_j} - \frac{\partial\overline{u_\iota u_J}}{\partial x_j} + S_{Ui}^{IF} \quad (3)$$

$$U_j\frac{\partial T}{\partial x_j} - \frac{\partial}{\partial x_j}\left(a\frac{\partial T}{\partial x_j}\right) = -\frac{\partial\overline{\theta u_J}}{\partial x_j} Y_d = e^{-(d/\bar{d})^n} \quad (4)$$

where U_i is averaged velocity component of the transport gas, ν is kinematic viscosity, ρ is density of transport gas, P is averaged pressure, $\overline{u_\iota u_J}$ are components of turbulent tensors, T is averaged temperature, $a = \lambda/\rho c_p$ is coefficient of heat diffusion and $\overline{\theta u_J}$ are the components of turbulent temperature fluxes.

Special attention should be dedicated to determination of the interphase members between primary and dispersed phase, due to the fact that presence of dispersed phase cause the appearance of new sources of momentum, mass and energy in primary phase. In majority of technical processes flows are usually multiphase, which complicates the solution of mathematical model. Also, it is important to emphasize that numerical modeling will be done with standard k -ε model of turbulence, which is most widely validated turbulence model.

Mathematical model based on Lagrangian approach includes tracking solid particle trajectories and on the basic of this concept, particle position, momentum, temperature, mass and interphase members along this trajectories can be determinated. This approach gives us much better, more realistic and more reliable picture of solid particle motion in turbulent flow. For fluid flows, that include lower concentration of dispersed phase governing equations can be formulated for each particle, but if the volumetric concentration of particles are bigger, statistical approach is much better, where tracking of each particle is changing with tracking of localized cloud of particles or parcels. Every parcel have the same mass, which is calculated as a product of individual mass of particle and their number in the parcel [24]. Parcels are injected in flow area uniformly at the inlet boundary with a velocity equal to the velocity of primary phase. Quantification of the dispersed flow in injected area, due to the previous research [1-21; 24-25], is done with Rossin-Rammler cumulative distribution function. For fully description of the same function several parameters from sieve analysis need to be defined. Rossin-Rammler distribution is based on the assumption that there is an exponential relation between particle diameter $\bar{d}$ and mass fraction of the particle with the diameter greater than $\bar{d}$, so the main equation have a shape:

$$Y_d = e^{-(d/\bar{d})^n} \tag{5}$$

where $\bar{d}$ is a middle diameter, and *n* is speeding parameter.

The basic description of parcel motion includes knowing the parcel position and their velocity. These two measures are connected with equation of motion, or trajectory equation:

$$\frac{dx_\pi}{dt} = U_p \tag{6}$$

where x_π is position vector U_p velocity of the particles (it is assumed that velocity of the particles are same as the velocity of parcel).

Due to the fact that we are considering chemically inert flow without any mass transfer, for fully description of motion of the dispersed phase, momentum equation is sufficient. So generic equation of motion for dispersed flow is defined as:

$$m_p \frac{dU_p}{dt} = F_S + F_b \tag{7}$$

where F_S refers to forces acting on the surface of particle, while F_b refers to mass forces.

If we consider the further decomposition of these forces, in our case, we will see that $F_S F_S$ is contained from drag force F_d and pressure gradient force, while F_b is contained only from gravity force. Of course, drag force is the main and dominant force which acts on the particles in the direction of flow and cause movement. Modeling of these force includes modeling of drag coefficient over various colorations based on theoretic and experimental investigations. As turbulence needs to be considered in modeling of primary flow, it also needs to be considered in modeling of secondary flow. Turbulent dispersion of solid phase was calculated with integration of trajectory of each particle separately, in which current particle velocity is used, along the whole trajectory. Calculation of trajectory for enough number of representative particles can take into account stochastic effect of turbulence on their dispersion. As it is previously mentioned, interphase members in basic governing equations represent the connection between primary and dispersed flow.

Due to the fact that in scope of this work momentum transfer between the flows is in the main interest, determination of momentum interphase members is put in the first place. Interphase member in momentum equation actually represents the drag force of solid particle in gas flow. In other words this force represents the resistant force to particle motion, which has an equal intensity but opposite direction from the drag force that causes the motion of particles. So interphase member that describes interaction between phases is determinated with solving of motion equation of the particle:

$$\frac{dm_pU_p}{dt} = S_{Ui}^{IF} \tag{8}$$

If we integrate the equation for every numerical cell and solve previous equation, we get an interphase momentum members for every numerical cell:

$$S_{Ui}^{IF} = \frac{\pi}{6}\sum\eta\left[\rho_p^0U_{p,i}^0\left(D_p^0\right)^3 - \rho_p^0U_{p,i}^n\left(D_p^n\right)^3\right] \tag{9}$$

where η is mass flow of particles for one cell, $U_{p,i}$ is component vector of particle velocity, D_p is diameter of particle, ρ_p is density of particle, n is end of Lagrange time step and 0 is start of Lagrange time step. If we assume that flow is stationary and that flow of particles that comes in control volume is equal to one that comes out, concentration of particles can be approximated on the basic of particle number that comes out from the control volume. So, particle flow is determinate with summarizing trajectories which cross observed control volume. [4, 17]

2.3. Boundary conditions

Due to the fact that determination of data for input boundary conditions and data for validation process were used from experimental measurements, it is necessary to properly describe where the same measurements were carried on. The whole system of mill-classifier (type S-36.50) and aero-mixture channels of boiler type OB 650 are very complex and measurement locations and measurement methods for data sampling are very limited.

In this study, aero-mixture channels one (1) and two (2) on the outlet of the classifier (Figure 3) were chosen as two representative measurement locations.

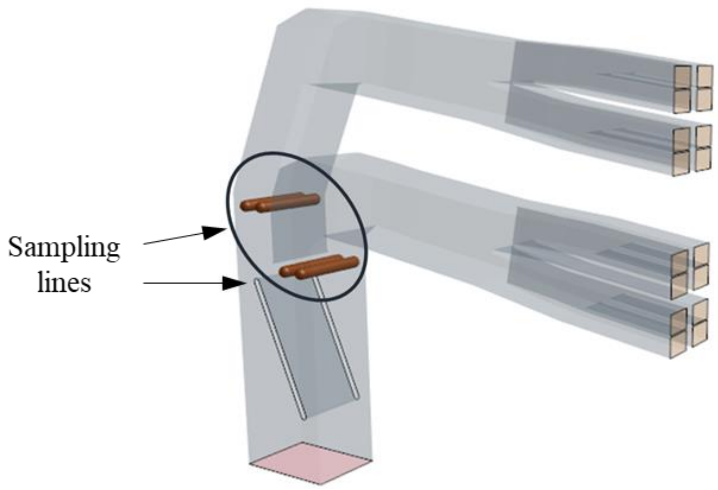

Figure 3. Measurement “line” locations on aero-mixture channel.

Every location have two measurement lines for sampling of gas velocity (Figure 3), particle velocity, their size, and distribution of both phases with all other specific parameters. Of course, it is important to emphasize that all these activities have been carried out under normal load conditions (6.83 kg/s of coal) and fixed positions of the manual regulation dumper with tilt of 23.52°. So, due to measurement possibilities and software capabilities, validation of gas velocity was done in sampling lines and validation of aero mixture distribution was done on the outlet of the jet burner, because the same doesn’t change after the regulation damper.

Previous general mathematical model that is used for description of problem requires the definition of appropriate boundary conditions that will approximate real state. In this case, because we have two coupled mathematical models, solving the same models requires the definition of boundary conditions for bots. So, in the next table, overview of used boundary conditions for gas phase is given.

Table 1. Used boundary conditions for primary flow.

Type of boundary from physical aspect	Type of boundary from mathematical aspect	Value of boundary condition
Inlet boundary	Dirichlet boundary condition	Inlet velocity v =23 [m/s], Inlet temperature t=195 [°C]
Outlet boundary	Dirichlet boundary condition	Pressure outlet $p = 0$ [Pa]
Wall boundary	Neumann boundary condition	Adiabatic boundary $\partial T/\partial n = 0$

Similar boundary conditions need to be defined, for dispersed particle flow. As it was previously defined in mathematical and physical model, for fully description of pulverized coal particle distribution and flow, Rosin-Rammler cumulative function is used. So from sieve analysis of the pulverized coal, for mill that has worked for 70 h, spreading parameter and middle diameter is determinated.

Table 2. Main Rosin-Rammler parameters.

Parameter	Middle diameter $\bar{d}$ [μm]	Spreding parameter n [/]
Value	154,554	1,4544

Of course, these parameters in table, determinate the granulometric structure of injected particle and they represent the first inlet boundary value for dispersed phase that needs to be determinated for further analysis.

When particle of dispersed phase comes to boundary, selected boundary condition defines the track of the same particle. So the main boundary condition that defines the track of the particles when particle comes to wall boundary condition is “rebound” boundary condition. With this boundary condition we define the change of momentum of the particles. The whole change of momentum is defined with two coefficients of restitution. First coefficient defines the change of momentum in normal direction and the second defines the change of momentum in tangential direction. These parameters have been adopted from previous researches [1-21], that have been taken into account the similar problematic of multiphase flow, so for value of the normal restitution coefficient is adopted value 1, and for tangential restitution coefficient 0.9.

The boundary condition defined on the outlet of the discretized domain for dispersed phase is “escape” boundary condition, due to the fact that on that boundary particles leave the domain and calculation of particle track is finished.

3. Analysis of obtained results and validation of numerical model

In the next step, with good approximation of real state and with good mathematical model and spatial discretization, we have a good predisposition to achieve quality results and their better and easier validation. Last step in process of numerical simulation is group of activity that implies calculation of numerical model on discretized area, and analysis of the solution. When numerical simulation reaches desired convergence of the main governing equations, a lot of spatial or plane views, of various scalar or vector values can be obtained. Of course, that was the main advantage of numerical modeling comparing to other methods. So in Figure 4, we have spatial distribution of pulverized coal particles in function of particle diameter.

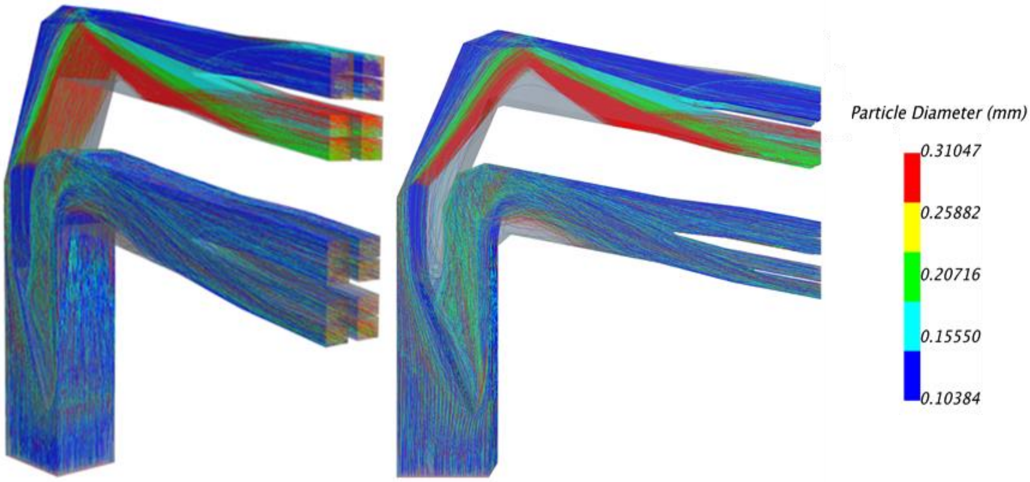

Figure 4. Spatial distribution of pulverized coal flow field in the function of diameter.

With aim to have better view in granulometric dispersion out of the burner in Figure 5. we have isolated detailed view.

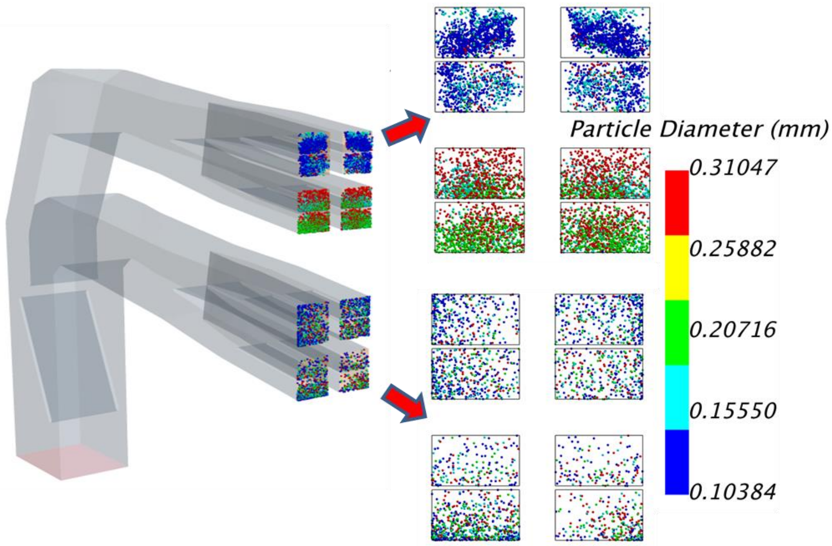

Figure 5. Detailed view of granulometric dispersion on outlet of the low emission burner.

Particle tracks in function of diameter give us only partial information about particle distribution inside the aero-mixture so velocity field of primary and secondary flow was shown in Figure 6.

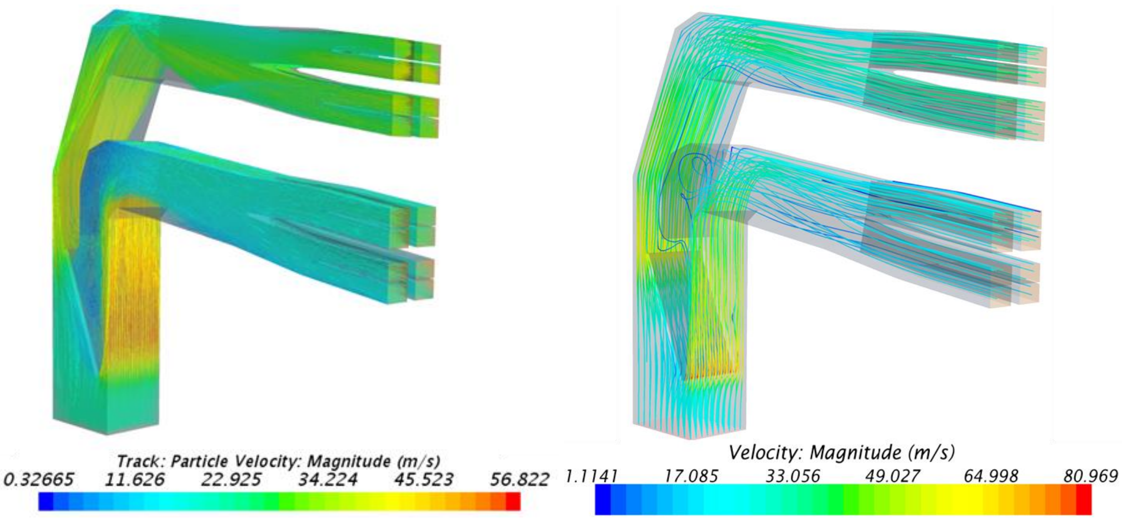

Figure 6. Spatial velocity field of pulverized coal (left). Spatial velocity field of primary phase (right).

In both figures we can see that dumper tilt has a big impact on structure of fields. This impact is reflected in the increase of velocity of primary and dispersed phase in upper part of aero-mixture channel, and decrease of velocity in lower part of aero-mixture channel.

A great advantage of numerical simulation is that even though we can obtain a lot of values in space, we can also quantified average values in certain plains or lines. So, in this case few averaged values on outlet planes and lines defined in Figure 1. and Figure 3. are tracked for validation of numerical model.

Value of every numerical model multiplies with validation of obtained results, with the values from experimental measurements. Validation of numerical model is done with experimental measurements when manual dumper is tilt for 23.52°.

Of course, matching values that we considered are velocity of primary flow in lines and separation of primary and dispersed phase on outlet of burner. Due to the diagram on the Fig.7, we can see extremely good matching of data, obtained from numerical simulation process, with data form experimental measurements, especially when matching of velocities and distributions of primary flow is in question, while coal distribution on the upper and lower part has lower deviations.

We can also see that deviations are insignificant and they can be seen especially in lower part of the burner. Obtained results have confirm that good numerical approximation of real state is done with this numerical model.

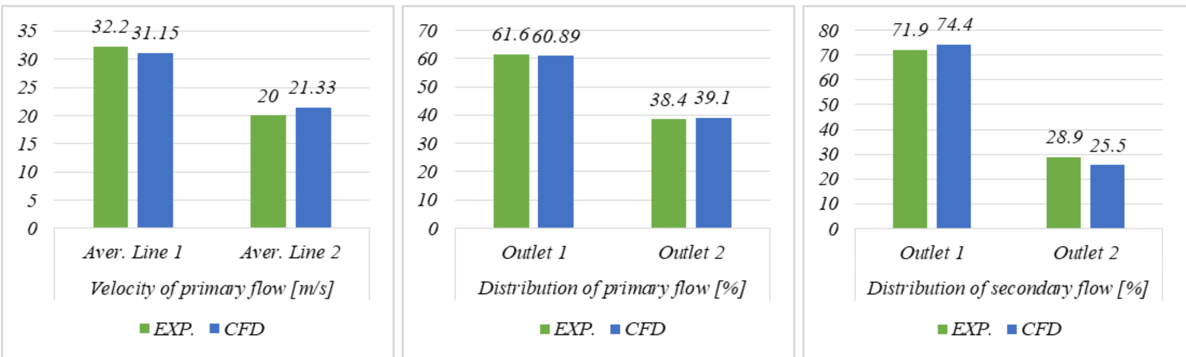

Figure 7. Validation of values obtained from numerical analysis

Proper impact of regulation dumper on aero-mixture distribution can be seen only with numerical simulation of aero-mixture channel with different regulation dumper positions. In this paper, an impact of regulation dumper on aero-mixture separation for four different positions was analyzed (Fig.8), from which the first was 23.52° (validated model-case no.1). The second was 0° (case no.2), in which regulation dumper has no angle. The third and the fourth were with angles ± 6.28° (case no.3 and no.4), regarding to validated model 23.52°.

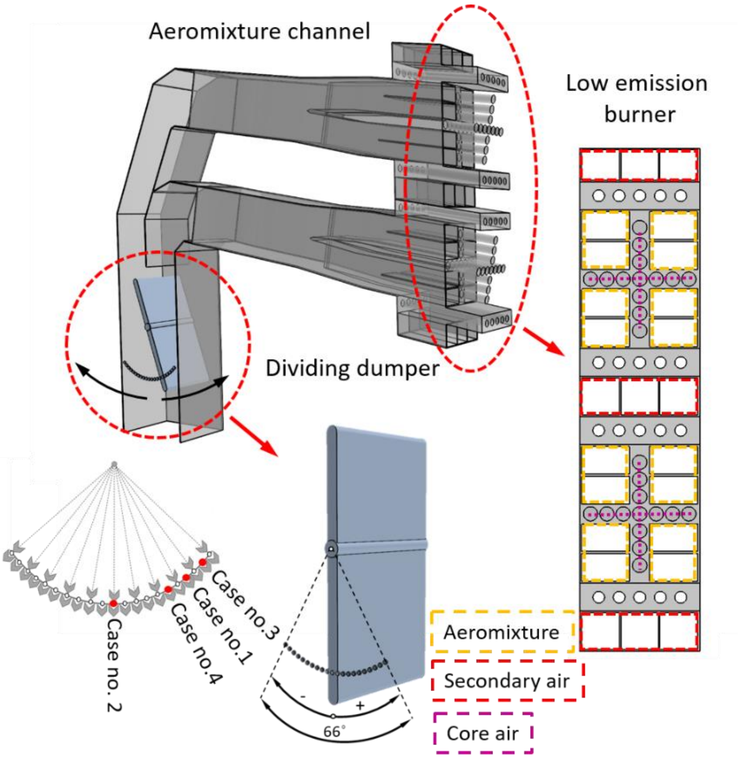

Figure 8. Four different analyzed positions of the manual dividing dumper.

Various conclusions about nature of dispersed particles along the furnace and nature of dispersed particles inside the channels can be obtained from numerical fields and quantified, average physics quantities on outlet of burner. Previous premise gives us space for additional geometric optimization of fluid flow which aims at the best and more uniformed dissipation of particles in aero-mixture channels. Same as it was done in previous cases, quantification was performed for average velocity values and for separation of primary and secondary phases on the outlet of the burner. For better interpretation of obtained results, values of previous physics quantities, in relation to angle of dividing dumper, is given in the next diagrams (Fig.9 and Fig.10).

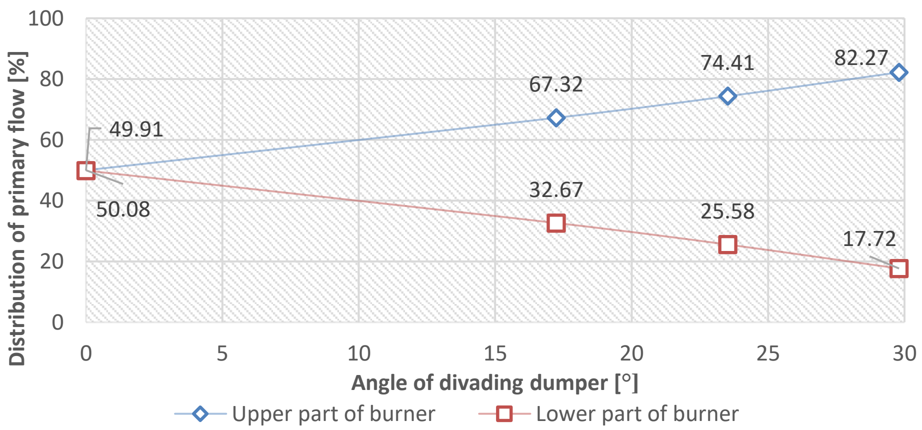

Figure 9. The dependency of angle of the dividing dumper on distribution of primary flow

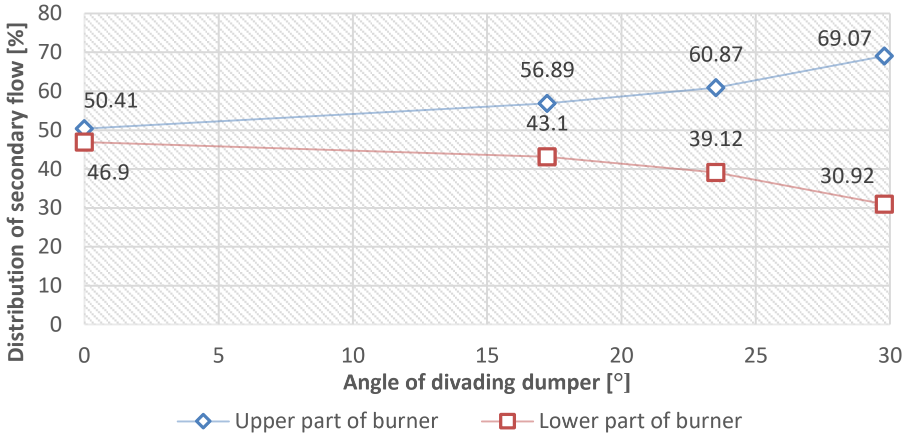

Figure 10. The dependency of angle of the dividing dumper on distribution of secondary flow

Diagrams on Fig.9 and Fig.10 obviously indicate that this regulation mechanism provides the opposite process in upper and lower parts of burner. Special attention in the diagram analysis, should be dedicated to values of velocity, due to the fact that the structure of the flame inside the combustion chamber depends from aero-mixture velocity, so in Fig.11 we have the dependency of angle of the dividing dumper on velocity of primary flow. Different values of kinetic energy on the outlet of upper and lower burner initiate opposite effects inside the combustion chamber, decreasing the penetration of coal particles into the flame.

On the other side, greater distribution of particles in upper part of combustion chamber has to insure greater residence time of coal particles in chamber space, which theoretically decreases loss of unburnt carbon in the slag.

Major step for the better understanding of multiphase fluid flow is an accurate insight in space and plane flow of primary and secondary phases inside the analyzed construction for different angles of manual regulation dumper. Certainly, better understanding of this problem provides us with different ways of the process regulation and management of aeromixture separation in practice.

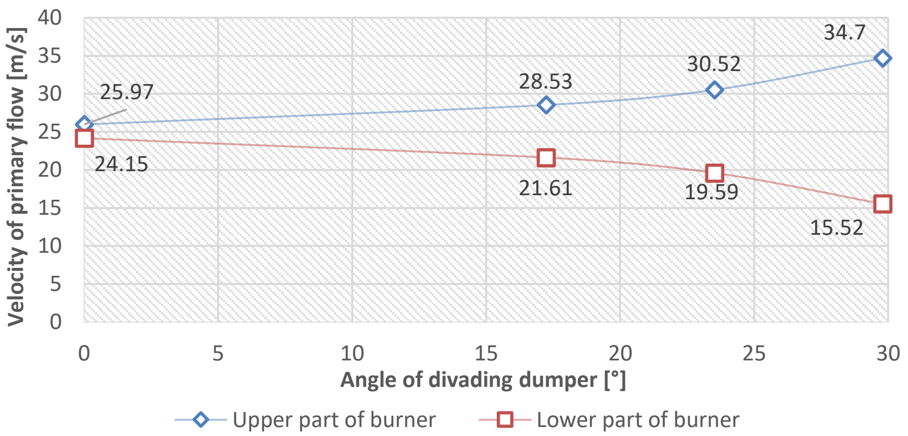

Figure 11. The dependency of angle of the dividing dumper on velocity of primary flow.

4. Conclusion

This numerical model, which mathematically describes multiphase flow inside the aero-mixture channel and low emission burner, represents the good numerical approximation of real state and gives us a view in spatial structure of certain fields that cannot be determinated with conventional methods. Good agreement between the simulated and experimental results was obtained for those results which are related to the gas, but a bit bigger deviation was detected in coal particle flow. Much complex determination of coal particle flow with classical measurement and its mathematical description is the main reasons for such deviation in results. We may conclude that this type of simulations is very useful in understanding the nature of the coal and gas flow rates in complex aero-mixture systems. This approach can help us to avoid complex and difficult measurements in some situations. Special significance of determination of the valid numerical model is reflected in fact that this kind of numerical model allows us applying of the same model on different working conditions, with different working parameters. Of course, obtained information's from that kind of models, about spatial and plane fields, have a particular importance in practice.

References

[1] G. STUPAR, D. TUCAKOVIC, T. ZIVANOVIC, V. IVANOVIC, D. KOMAROV: 3D Model Of Solid And Gas Phase Flow In The Duct Bend Behind The Mill Gas Classifier At The Fan Mill. In: 24th International Conference on Efficiency, Cost, Optimization, Simulation and Environmental Impact of Energy Systems, *Novi Sad,* July 2011, 786-797. DOI: 10.13140/2.1.4276.2565

[2] M. KOZIĆ, M. PUHARIĆ, S. RISTIĆ, B. KATAVIĆ: Numerička simulacija strujanja u ventilacijskom mlinu i kanalu aerosmješe termoelektrane Kostolac B, *Strojarstvo*, vol.53, no. 2, pp. 83-90, 2011. https://hrcak.srce.hr/76094

[3] M. KOZIĆ, S. RISTIĆ, M. PUHARIĆ, B. KATAVIĆ: Comparasion of Euler-Euler and Euler-Lagrange approach in numerical simulation of multiphase flow in ventilation mill-air mixing duct. In: Proceedings of the Third Serbian Congress on Theoretical and Applied Mechanics. *Vlasina lake*, 2011, 290-303. http://ssm.org.rs/archive/Congress2011/Proceedings/4_Section%20B_Part2.pdf

[4] S. MILANOVIĆ, M. JOVANOVIĆ, Ž. SPASIĆ, B. NIKOLIĆ: Two-phase flow in channels with non-circular cross-section of pneumatic transport of powder materia. *Thermal Science*. 22 (5), 1407-1424, (2018). https://doi.org/10.2298/TSCI18S5407M

[5] G. ŽIVKOVIĆ, S. NEMODA, P. STEFANOVIĆ, P. RADOVANOVIĆ: Numerical analysis of the influence of louvers on the coal powder distribution in boiler burning channels on TENT-A6. *Termotehnika*, ISSN 0350-218X, **34** (2-3), 133-145 (2008). https://scindeks.ceon.rs/article.aspx?artid=0350-218X0803133Z

[6] S. ATAS, U. TEKIR, MA. PAKSOY, A. CELIK, M. CAMA, T. SEVGEL: Numerical and experimental analysis of pulverized coal mill classifier performance in the Soma B Power Plant. *Fuel Processing Technology Journal*, ISSN 0378-3820, **126**, 441-452, (2014). DOI:10.1016/J.FUPROC.2014.05.016

[7] M. KOZIĆ, S. RISTIĆ: CFD Analysis of the Influence of Centrifugal Separator Geometry Modification on the Pulverized Coal Distribution at the Burners. *Transactions of FAMENA*, **38** (1), 2014. https://hrcak.srce.hr/120154

[8] N. ŽIVKOVIĆ, G. ŽIVKOVIĆ, P. STEFANOVIĆ, D. CVETINOVIĆ: Numerical analysis of the flue gas-coal particles mixture flow in burner's distribution channels with regulation shutters at the TPP Nikola Tesla - A1 utility boiler**.** *Thermal Science*. 14 (2), 505-520, (2010). https://doi.org/10.2298/TSCI1002505Z

[9] V. BABIĆ, T. ŽIVANOVIĆ, Ž. STEVANOVIĆ, Z. MILOVANOVIĆ: Numerical simulation of the flow in coal powder inertial separator. Energy efficiency and effective operation of power plants. In: Proceedings of the International Conference on Thermal Power Plant, *Zlatibor*, 2012, 982-990.

[10] J.L. FERRIN, L. SAAVEDRA: Distribution of the coal flow in the mill-duct system of the As Pontes Power Plant using CFD modelling. *Fuel Processing Technology Journal*, ISSN 0378-3820, **106**, 84-94 (2013). https://doi.org/10.1016/j.fuproc.2012.07.005

[11] D. DODDS, J. NASER, J. STAPLES, C. BLACK, L. MARSHALL, V. NIGHTINGALE: Experimental and numerical study of the pulverised-fuel distribution in the mill-duct system

of the Loy Yang B lignite fuelled power station. *Powder Technology*, ISSN 0032-5910, **207** (1-3), 257–269, (2011). https://doi.org/10.1016/j.powtec.2010.11.007
[12] H. VUTHALURU, VISHNU PAREEK, R. VUTHALURU: Multiphase flow simulation of a simplified coal pulveriser. *Fuel Processing Technology Journal*, ISSN 0378-3820, **86** (11), 1195–1205, (2005). https://doi.org/10.1016/j.fuproc.2004.12.003
[13] C. ARAKAKI, A. GHADERI, A. SÆTHER, C. RATNAYAKE, G.G. ENSTAD: Air mass balance for mass flow rate calculation in pneumatic conveying. *Powder Technology*, ISSN 0032-5910, **202** (1–3), 62–70, (2010). https://doi.org/10.1016/j.powtec.2010.04.007
[14] M. VUJANOVIĆ: Numerical modelling of multiphase flow in combustion of liquid fuel. *PhD thesis*. University of Zagreb, Faculty of Mechanical Engineering and Naval Architecture; 2010. https://urn.nsk.hr/urn:nbn:hr:235:036060
[15] M. KOZIC, S. RISTIC, B. KATAVIC, M. PUHARIC: Numerical simulation of multiphase flow in ventilation mill and channel with louvers and centrifugal separator. *Thermal Science* **15** (3), 677-689, (2011). https://doi.org/10.2298/TSCI101203018K
[16] N. CRNOMARKOVIĆ: Contribution to the modeling of the spatial distribution of radiation inside the furnace of the pulverized coal fired boiler. *PhD thesis*. University of Belgrade, Faculty of Mechanical Engineering, 2012. 10.2298/BG20120921CRNOMARKOVIC.
[17] T. NIEMI: Particle Size Distribution in CFD Simulation of Gas-Phase Flows. *Master thesis*, Aalto University, School of Science; *Aalto, Finland*, 2012.
[18] S. BELOŠEVIĆ, I. TOMANOVIĆ, V. BELJANSKI, D. TUCAKOVIĆ, T. ŽIVANOVIĆ: Numerical prediction of processes for clean and efficient combustion of pulverized coal in power plants. *Applied Thermal Engineering*, **74**, 102-110, (2015). https://doi.org/10.1016/j.applthermaleng.2013.11.019
[19] A. RASEL, M. SULTAN, AZIZ RAHMAN, SAYEED RUSHD, SOHRAB ZENDEHBOUDI, C. VASSILIOS: Validation of CFD model of multiphase flow through pipeline and annular geometries, *Particulate Science and Technology*, **37**(6), 685-697, (2019). https://doi.org/10.1080/02726351.2018.1435594
[20] L. KRANJČEVIĆ, Z. ČARIJA, S. FUĆAK: Numerical mesh impact on computational simulation efficiency. *Engineering review,* **27** (2), 25-36, (2007). https://hrcak.srce.hr/26338
[21] CD-adapco. Star CCM+ - *User Guide*, 2014.
[22] Ansys 15.0 – *User Guide*, 2013.
[23] I. DŽIJAN: Računalna dinamika fluida. Faculty of Mechanical Engineering and Naval Architecture, *Zagreb*, 2010.
[24] N. BARBALIĆ, E. DŽAFEROVIĆ: Transport čvrstih čestica fluidom. Faculty of Mechanical Engineering, University of Sarajevo, *Sarajevo*, 2006.
[25] T. ŽIVKOVIĆ, LJ. BRKIĆ, D. TUCAKOVIĆ: Proračun postrojenja za pripremu ugljenog praha, Faculty of Mechanical Engineering, University of Belgrade, *Belgrade*, 2005.