

Application of Computational Fluid Dynamics to optimization of cyclone dust separators operated in the cement industry

Marek WASILEWSKI, Jerzy DUDA – Faculty of Production Engineering and Logistics, Opole University of Technology, Poland

Please cite as: CHEMIK 2013, 67, 10, 985–994

1. Introduction

Cyclones are the mechanical dust separators that are the most frequently used for technological processes to extract distributed solid particles for a liquid carrier. They found application in various industrial sectors, including power engineering, chemistry, manufacturing of commodities, timber processing as well as for production of cement. Such equipment features with a simple design, low cost of manufacturing and maintenance and nearly unmanned operation. In addition, they perfectly perform under heavy-duty conditions with wide range of temperature and pressure, which makes it possible to customize typical units for needs of specific plants. Efficiency of dust separation achieved by cyclones may reach up to 97% for particles of the size above 10 μm , even under high content of solids in the input feed of liquid. Cyclone separators are applied to extraction of large particles for the purpose of control contamination of air as well as for smooth course of technological processes. Application to harsh conditions includes such processes as separation of coal dust downstream coal pulverizers at power plants or dehydration of materials at drying systems. They can be also used as gas reactors or heat exchangers.

Operation principle of cyclone dust separators consists in employment of centrifugal force cause by vortex movement of liquids. The multi-phase blend of fluid and solids is supplied to the upper part of a cyclone. Vortex flow of the blend through the cyclone leads to concentration of the solid phase nearby walls of the outermost cylinder. Since the descending spiral shape of the flow channel the deposited solids are transferred downwards to the discharge port. In turn, the fluid phase is reversed and then transferred upward where it is released outside the unit through outlet channel aligned with the central axis of the unit.

Key parameters that determine efficiency of cyclone separates are dust separation efficiency and pressured drop. These parameters depend on geometrical parameters of each cyclone and operating conditions. Exact determination of these parameters is extremely important since they are crucial for cost efficiency of the process and, in direct consequence, operation expenses. Higher discharge velocity improves efficiency of solid separation but paid by higher pressure drops. Thus, the compromise must be achieved between maximum achievable dust separation efficiency from the separation process and the minimum pressure drop across the cyclone.

Among various technologies that are used for clinker burning the least power consuming is the dry method that is commonly used by most of cement plants worldwide. The modern clinker production systems comprises a rotary kiln, the suspension preheater and a calcinator. One of solution that improves thermal efficiency of the burning process is optimization of the cyclone heat exchanger (suspension peheater). Its efficiency of dust separation and utilization of heat substantially depends on the applied cyclones, number of their stages and it also affects thermal efficiency of the kiln. Refurbishment of existing heat exchangers entails huge financial lost due to the need to stop operation of the kiln for long downtime and high investment expenses. Therefore alternative and simpler solutions are sought to enable operation of the heat exchanger with the minimum downtimes of the kiln.

Operation of the Suspension Preheater (SP) is similar to conventional heat exchangers that operate with suspensions. Figure 1 explains operation principle of such exchangers while the detailed description of the operation cycle is provided in [1, 2].

With consideration to sophisticated dynamic features of processes that run inside cyclones all experimental investigations intended to improve operation efficiency of the units are time consuming and expensive. The solution for these problems can be found in application of a mathematical model for investigation and prediction of flows for various geometrical designs of cyclones. It turned out that the effective tool for investigation of cyclone characteristics can be Computational Fluid Dynamics (CFD).

The major objective of this study was application of the CFD method to find out how shapes of characteristic key components of cyclones affect performance parameters and behaviour of the flow field in order to optimize design of cyclones that are involved in the process of clinker burning.

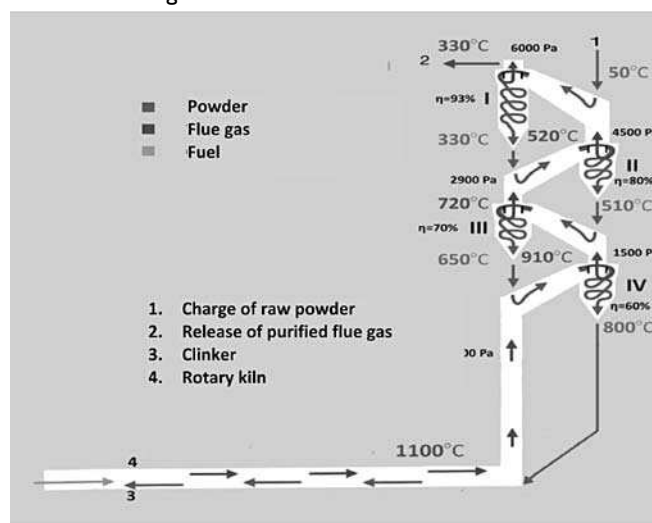


Fig. 1. Operation principle of a cyclone Suspension Preheater

2. Numeric modelling of the blend flow throughout a cyclone Suspension Preheater

Computational Fluid Dynamics (CFD) is a universal tool that enables prediction of characteristics for flow fields, trajectories of particles inside a cyclone and pressure drop. The sophisticated vortex and irregular flow of fluid inside the cyclone pose a real challenge to numerical technologies and development of models for turbulences used for CFD codes.

The investigations were focused on a model for the 1st stage of a cyclone (Fig. 2) that is in service in one of cement plants in Poland. The cyclone features with a modern design (very high separation efficiency with the minimum possible hydraulic resistance). The unit is incorporated into the entire tower of heat exchangers that is made up of four stages. The stages from 2nd to 4th are designed as dual systems whilst the 1st stage comprises four cyclones. The number of cyclones for individual stages correspond to overall productivity of the manufacturing plant.

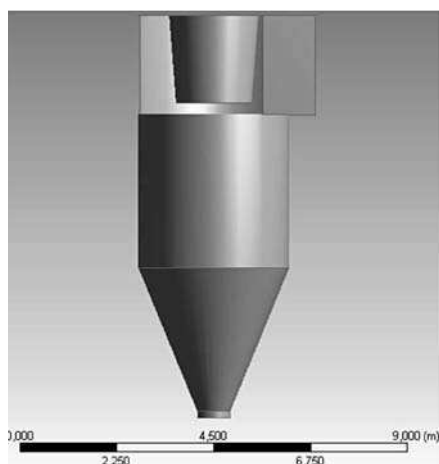


Fig. 2. Optical model for the 1st stage of the cyclone dust separator

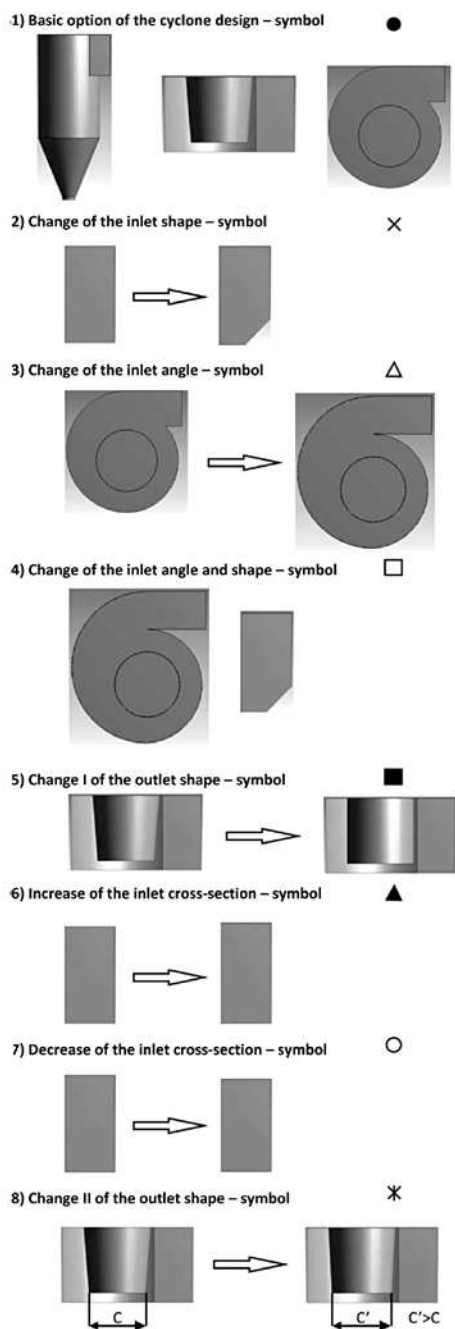


Fig. 3. Options for modification of the cyclone design used for investigations

The operating conditions of the unit were determined on the basis of the mass and heat balance, kiln productivity, amount of flue gas, distribution of temperatures inside the suspension preheater and average results from measurement taken during the plant operation (including the mass flow of feed and gas) (Tab.1).

Size of solid particles supplied to a cyclone is an important parameter for the cyclone design. Therefore it is first necessary to really accurately determine distribution of solids. Due to the fact that the solid phase – the raw material in the form of powdered mixture – comprises particles of various sizes, shares of individual size fractions had to be determined at first. Execution of the screen analysis made it possible to determine percentage shares of individual fractions (in total 18 fractions were identified) under the assumption of the Rosin-Rammler-Sperling (RRS) distribution.

Table I

Operating conditions assumed for investigations of feed flow

Solid phase – CaCO ₃	
Density	2800 kg/m ³
Range of the particle size	5 μm to 500 μm (18 fractions)
Mass flow	16.7 kg/s
Gaseous phase – air	
Density	1.225 kg/m ³
Mass flow	64.7 kg/s

Figure 3 presents possible options for modification of the cyclone design.

The analysis comprises the basic option (marked as No. 1) and seven (7) modifications (numbered from 2 to 8). For clear illustration of results each modification is provided with a corresponding symbol. Selection of modification options was done through analysis of solutions described in the literature, which enables determination of such typical components of the dust separator that the most crucial for the device operation.

2.1. Theoretical fundamentals for numerical modelling

Computational Fluid Dynamics (CFD) is the method for simulation of systems, processes and equipment that are involved in flow of fluids, exchange of heat and mass, chemical reactions and similar physical phenomena. It is achieved by numerical resolution of differential equations that describe behaviour of momentum, mass and heat, so called mathematical models. Application of the CFD method makes it possible to carry out the detailed analysis of issues associated with flow of fluids with simultaneous elimination of the need to carry out time consuming and expensive experimental test, normally unavoidable for the cycle of equipment engineering and refurbishment. CFD software offers the possibility to acquire necessary data for fluid flow (e.g. distribution of velocity and/or pressure fields), heat transfer (temperature field) and other associated phenomena (including chemical reactions). The gathered information enable initial qualitative analysis and further more detailed quantitative interpretation of obtained results.

Taking account of the fact that the equations for conservation of momentum, mass and energy comprise identical structures that describe contribution of specific mechanism of transportation, the following equation can be adopted:

$$\frac{\partial \rho \phi}{\partial t} + \text{div}(\rho \phi \vec{w}) = \text{div}(\Gamma \text{grad} \phi) + S_{\phi} \quad (1)$$

where:

ρ – density, kg/m³

t – time, s

\vec{w} – averaged velocity vector, m/s

S_{ϕ} – source segment,

Γ – diffusion factor

ϕ – generalized scalar variable (results from identical of segments)

Owing to the possibility to write all transfer equations in the form of a single equation only one numerical procedure shall be necessary that shall be repeated for appropriate ϕ (and corresponding expressions for Γ and S_{ϕ})

Predominant majority of flows that occur in chemical and process engineering is of turbulent nature and such flows exhibit irregularity, vortex phenomena diffusive properties and discontinuity. It is why for CFD methods must enable modeling of phenomena associated with

turbulent flow. For that purpose the method of Reynolds-Averaged Navier-Stokes Equations (RANS) is used for most applications. The Navier-Stokes (N-S) equation adopts the following form (the equation of momentum conservation):

$$\frac{\partial(\rho\bar{w})}{\partial t} + \text{div}(\rho\bar{w}\bar{w} + \rho\overline{w_i'w_j'}) = \text{div}(\mu\text{grad}\bar{w}) - \text{div}(p\vec{I}) + \rho\vec{g} \quad (2)$$

(i, j, k = 1, 2, 3 or x, y, z stand for vector components in the Cartesian coordination system).

Appearing of the new module $\rho\overline{w_i'w_j'}$ leads to the situation that the equation system for description of turbulent flows becomes a non-isolated system.

Turbulent stress is presented in the following form:

$$-\rho\overline{w_i'w_j'} = \mu_T \left(\frac{\partial\bar{w}_i}{\partial x_j} + \frac{\partial\bar{w}_j}{\partial x_i} \right) - \frac{2}{3}\rho\delta_{ij}k \quad (3)$$

where:

μ_T – turbulent viscosity, Pa*s

δ_{ij} – Kronecker delta function

$\bar{w}_i, \overline{w_i'}$ – averaged velocity vector and its fluctuation (the tensor notation of the x_i direction)

k – kinetic energy of turbulences (normalized for a single unit of the fluid weight)

Resolution of such a problem needs formulation of a closure hypothesis, so called detailed model. In case of cyclone dust separators the closure model that is used in most cases assumes the *Reynolds Stress Model* (RSM). It is the model that enables accurate mapping of turbulent stress of the highly anisotropic nature (that actually occurs in cyclones) and was described with details in [3÷5].

To evaluate efficiency of solid separation inside cyclones one can use the Euler-Lagrange approach (EL) as the *Discrete Phase Model* (DPM). The methodology assumes that the fluid is considered as a continuous phase (the computations consist in resolving of the Navier-Stokes equation), whilst the distributed phase (suspended matters) is identified by tracking of particles (calculation of the flow field). Trajectories for individual groups of particles are predicted by execution of the balance for forces that act on each specific group of particles. The detailed description of the foregoing method is provided in [6, 7].

2.2. Methodology of investigations

The numerical analysis of flows was carried out by means of the Finite Volume Method with use of the Fluent software package from ANSYS Inc. To resolve differential equations the SIMPLE computation algorithm was applied with the aim to correctly determine the coupling between the field of pressures and velocities for the course of the equation for conservation of momentum. The model consists in alternate computations of iterative approximations for velocity and pressure components [8, 9]. To find out representative samples for component values on surfaces of control volumes the computations employed the First-Order Upwind method. The convergence condition for the continuity equation was set as 10^{-5} and for the rest of equations was 10^{-3} . For modeling of the boundary layer the standard wall-adjacent functions were defined. In addition, standard values of subrelaxation coefficients were used to stabilize the computation process.

The efforts were commenced with development of three dimensional geometrical models that corresponded to every option presented in Figure 3. Due to geometrical properties of the apparatus under test the 3D solver was applied. Then the developed models were imported to the Ansys Meshing software that was used for their discretization. The numerical mesh had cells of hexagonal shape and its density varied depending on location within the apparatus (the highest density in the areas of the cyclone inlet and outlet, the lowest density in conical parts). The total number of cells varied between 400,000 and 420,000 depending on the option of the cyclone design that was subjected to the analysis. It was the minimum number of cells that made it possible to avoid substantial impact of the mesh density growth to the final result. Simulations were carried

out in the computation domain that corresponded to full geometry of the simulated problem, i.e. without use of symmetry planes.

Due to the fact that selection of the adequate model of turbulences is the matter of key importance, the detailed investigation of reference literature was carried out. It was found out the k-ε model, although the most popular, is not sufficiently accurate for simulation of flow inside cyclones due to isotropic nature of that model in case of turbulences [10÷13]. Instead, the decision was taken to apply the Reynolds Stress Model (RSM). It is the model that requires resolution of transportation equations for each component of the Reynolds stress, which enables more detailed description of flow phenomena (including characteristics of vortexes, axial and tangential velocities, pressure drops, i.e. the parameters that are extremely important for modeling of flows in cyclone dust separators). It is also the model that makes it possible to accurately reproduce turbulent stresses of highly anisotropic nature.

Presence of the solid phase was modeled by means of the Euler-Lagrange method, which was implemented into the ANSYS FLUENT software as a Discrete Phase Model (DPM). It is the methodology that considers fluid as a continuous phase (computations consist in resolving of the N-S equation), whilst the distributed phase is identified by tracking of particles (calculation of the flow field). Trajectories for individual groups of particles are predicted by execution of the balance for forces that act on each specific group of particles. Such an approach enables assessment of the cyclone efficiency. The computations took also account of interactions between the continuous and distributed phase.

Prior to commencement of analysis for individual options of the cyclone modification the results obtained from the calculations for the basic option (I) were validated against values obtained from measurements of working parameters during the plant operation. For pressure drops the conformity was 89% and for dust separation efficiency the computations and actual results matched in 96%. Such a conformity enables to judge that the applied model of turbulences can be successfully applied to simulation and visualization of gas-solid blends that flows through cyclone separators.

2.3. Results and analysis of computations

Figure 4 presents values of pressure drop and dust separation efficiency for the options of the cyclone modification.

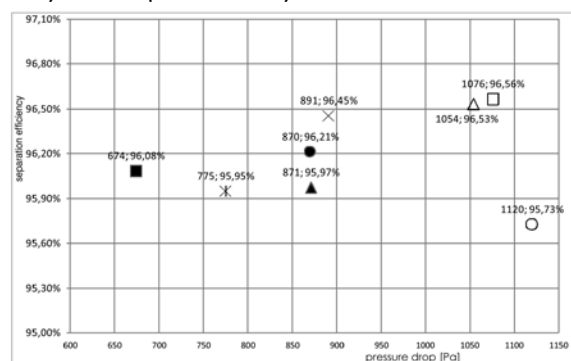


Fig. 4. Separation efficiency and pressure drop values for various option of modification

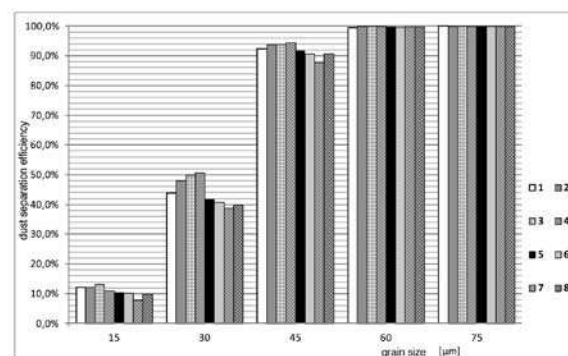


Fig. 5. Dust separation efficiency for various design option and grain size <75 μm

Total pressure loss is defined as the difference between average static pressure at the cyclone inlet and its outlet. The separation efficiency is meant as the ratio between the weight of particles captured by the device to the total mass flow of solids.

Depending on the objective function assumed for the optimization the achieved results can be considered in two manners:

- when the objective function is defined as efficiency of dust separation the option No. 4 proved to be the optimum solution (the least amount of the material feed escapes outside the system).
- for minimization of pressure drops the best solution is represented by the option No. 5 (less consumption of electric power for driving motors of fans).

With regard to efficiency of solid separation it was spotted that for particles with the diameter of $75\mu\text{m}$ and more the dust

separation efficiency is 100% for all design options. Therefore Figure 5 shows dust separation efficiency only for the fraction of particle size below $75\mu\text{m}$.

To better understand how stream of particles flows inside the cyclone, Figure 7 depicts contour magnitudes of velocities. One can spot that velocity of internal flow increases down the central line from the cyclone bottom to its top. The feed velocity at the inlet considerably affect the number of turns in the inner vortex but its effect onto the outer vortex is less significant. In addition, the vortex length and backflow increase in pace with the inlet velocity.

In turn, Figure 8 shows examples for distribution of static pressure on longitudinal cross-sections of cyclones. One can easily see that the zone of low pressure is in the centre of the dust separator and then the static pressure grows down the radial direction

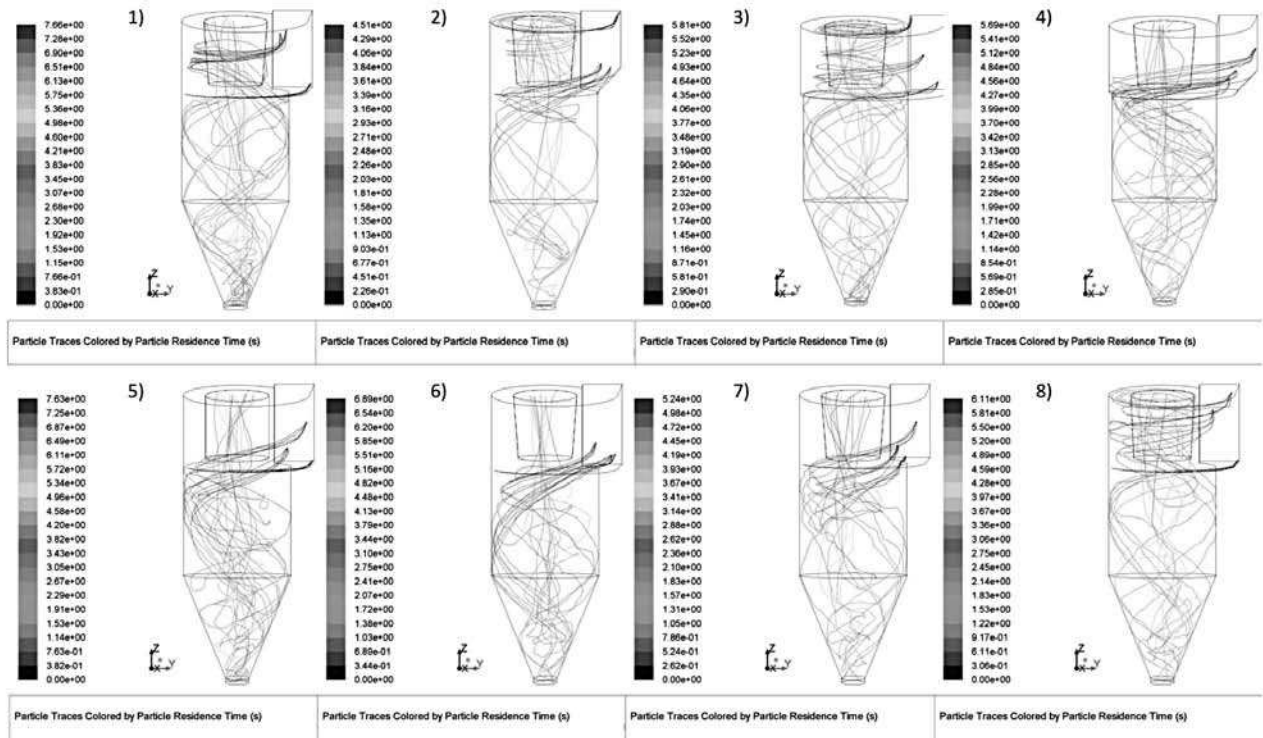


Fig. 6. Trajectories of solid particles with the size of $15\mu\text{m}$

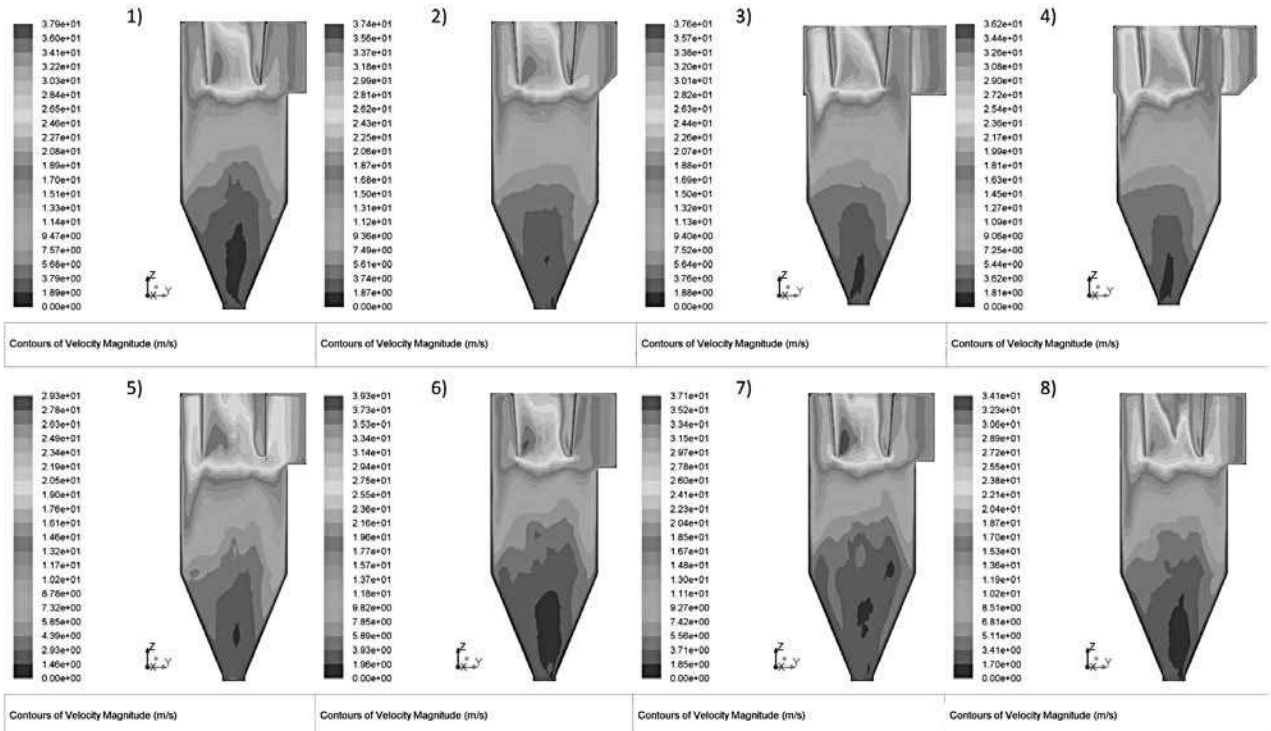


Fig. 7. Contours of velocity magnitudes for individual design options

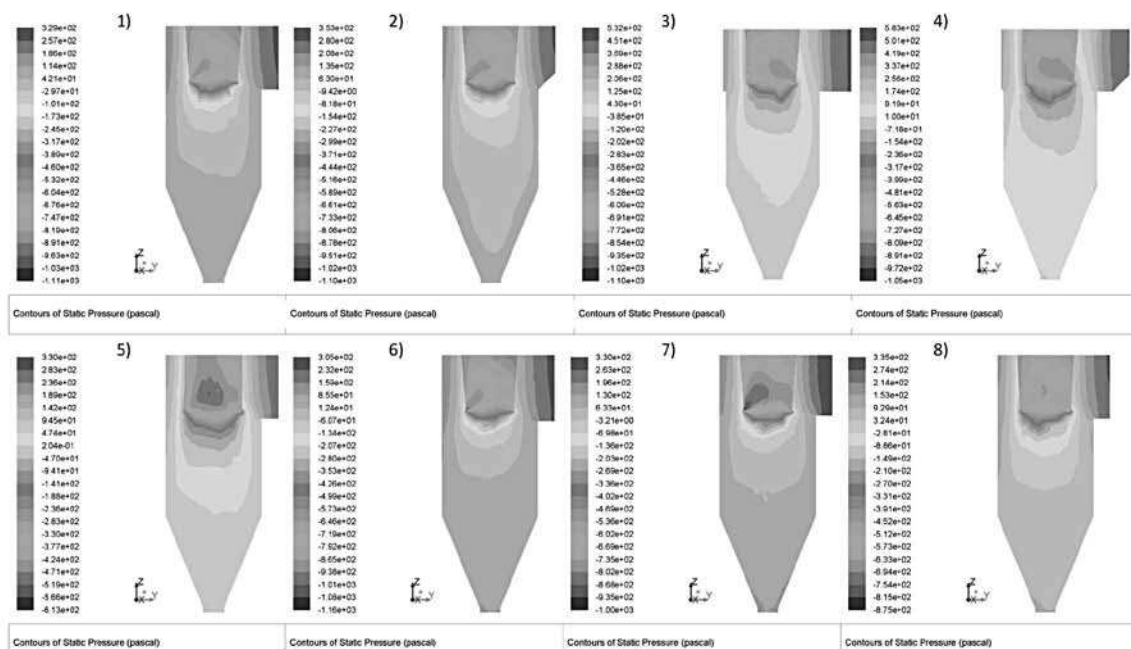


Fig. 8. Distribution of static pressure on longitudinal cross-sections

3. Recapitulation and conclusions

Whilst resolving of numerical problems associated with flow phenomena one has to remember that no general model of flow exists and no model is able to guarantee correct results for various classes of flows since every model is based on some simplifying assumptions. Prior to application of any specific model it is necessary to survey available solutions from referenced literature and achieved results should be compared against experimental results (model validation). The disclosed computations base on the RSM model of turbulences that is described in relevant literature as the most suitable for cyclone dust separators since it takes account of the phenomena associated with the flow of anisotropic nature. The completed validation (for the option No. 1) makes it possible to confirm that the assumed mathematical model can be successfully applied to visualize flow of fluid and solid blends in cyclones (results from numerical computations demonstrate high convergence to figures measured for a dust separation system in service).

The completed analysis enabled interpretation of measurement data and find out which design of the cyclone structure is the most beneficial for the assumed flow of fluid and solids. Familiarity with flow processes inside a cyclone dust separator may serve as the starting point for optimization of its design. Review of possible alterations to the cyclone design demonstrated that the option No. 4 proved to be the most beneficial in terms of dust separation efficiency. However, when minimization of flow resistance is the point, the option No. 5 seems to be the best. In practice, the real objective is a reasonable compromise between high efficiency of separation and low pressure drop across the device. If so, the most favourable solution is the option No. 2.

Further investigation assume that a series of experiments shall be carried out in order to confirm computations for the assumed design options against results of measurements.

Literature

1. Duda J.: *Badania izotopowe przepływu materiału w nowoczesnym piecu obrotowym pracującym metodą suchą*. Prace IMMB 1994, **13**, 105–112.
2. Wasilewski M., Duczkowska-Kądziel A., Duda J.: *Wykorzystanie nowoczesnych technik modelowania numerycznego w celu minimalizacji spadku ciśnienia w odpylaczach cyklonowych stosowanych w procesie wypalania klinkieru*. Innowacje w zarządzaniu i inżynierii produkcji 2013, t. II, część 5, 687–696.
3. Su Y., Zheng A., Zhao B.: *Numerical simulation of effect of inlet configuration on square cyclone separator performance*. Powder Technology 2011, część **210**, 293–303.
4. Wan G., Sun G., Xue X., Shi M.: *Solids concentration simulation of different size particles in a cyclone separator*. Powder Technology 2008, część **183**, 94–104.

5. Wang B., Xu D.L., Chu K.W., Yu A.B.: *Numerical study of gas-solid flow in a cyclone separator*. Applied Mathematical Modelling 2006, część **30**, 1326–1342.
6. Ferziger J.H., Peric M.: *Computational Methods For Fluid Dynamics*. Springer 2002.
7. Dobrowolski B., Nowosielski G.: *Modelowanie zjawisk przepływowych w odsiewaczu młyna wentylatorowego*. Inżynieria Chemiczna i Procesowa 2004, **25**, 2097–2194.
8. Gryboś R.: *Podstawy mechaniki płynów*. PWN, 1998.
9. Pantakar S.V.: *Numerical heat transfer and fluid flow*. Hemisphere Publishing Corporation, Washington, 1980.
10. Cristea E.D., Malfa E., Coghe A.: *3-D Numerical simulation and measurement of strongly swirling heavy dust-laden flow inside a cyclone separator*. 3rd International Symposium on Engineering Turbulence Modelling and Measurements, May 27–29 1996, Crete.
11. Meier H., Mori M.: *Anisotropic behavior of the Reynolds stress in gas and gas-solid flows in cyclones*. Powder Technology 1999, część **101**, 108–119.
12. Kozołub P., Adamczyk W., Białecki R., Węcel G.: *Numerical estimation of the particle collection efficiency of a cyclone separator*. XX Polish Fluid Mechanics Conference, Gliwice, 2012.
13. Parihar A.K.S., Joshi Ch., Sridhar G.: *The performance of cyclones in producer gas cleaning: experimental and modeling studies*. Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy 2012, 1–18.

Marek WASILEWSKI – M.Sc., graduated from the Faculty of Mechanical Engineering at the Opole University of Technology. Currently he is employed as an assistant in the Chair of Innovative Technological Processes, Faculty of Production Engineering and Logistics, Opole University of Technology. His scope of research interests include process engineering, environmental engineering, optimization of manufacturing processes with particular attention to manufacturing of cement. He is a scientific editor of one monograph book, author of 5 chapters in monograph books and two papers and posters at national and international conferences

e-mail: m.wasilewski@po.opole.pl, tel.: +48 77 449 88 49

Jerzy DUDA – (Sc.D., Eng.), Professor, graduated from the Faculty of Mechanical and Power Engineering at the Wrocław University of Technology. He presented his PhD thesis in 1985 at the Institute of Thermal Technology and Mechanics of Fluids at the Wrocław University of Technology. He was nominated for the professor position after presentation of Sc.D. thesis in 2006 at the Mechanical and Power Engineering Faculty at the Wrocław University of Technology. Since 2007 he has been employed as titular professor at the Opole University of Technology. Currently heads the Chair of Innovative Technological Processes, Faculty of Production Engineering and Logistics, Opole University of Technology. He published more than 110 manuscripts and papers, received 11 patent certificates and registered 2 utility patterns. His scientific output counts more than 100 research programs with splendid result that have been implemented in great deal in the cement and limestone industries. The scope of his research interests include heat generation and power engineering, optimization of power generation processes, application of renewable energy to technological processes.

e-mail: j.duda@po.opole.pl, tel.: +48 77 449 88 47