SAT-9.3 -1-HP-14

CFD STUDY OF TWO-PHASE FLOW BEHAVIOR IN THE CYCLONE. EFFICIENCY ANALYSIS[141]

Angel Terziev, Assoc. Prof., PhD
Assoc. Prof. Angel Terziev, Dept. of "Hydroaerodynamics and hydraulic machines", Technical University of Sofia, Phone (02) 965 3443,
E-mail: aterziev@tu-sofia.bg

Abstract: *Current paper focuses on the CFD study of the behaviour of the two phase flow (processed exhaust gas flow carrying solid particles) in an industrial cyclone. An analysis of the velocity and temperature fields of the two phase flow at different geometrical parameters and inlet flow data was presented. Based on the accepted initial condition a regression analysis of the efficiency of the cyclone was made and significant parameters were selected.*

Key words: *CFD, cyclone efficiency, two phase flow behavior.*

INTRODUCTION

Most industrial processes are related with the emission of harmful substances in working premises. Very often these substances are solid particles with reasonable parameters and based on current environmental norms it is not allowed to be freely released into the environment. Different types of systems are used to "catch" the solid particles and to evacuate them away from the working premises. The exhaust air carrying solid particles is actually two—phase flow. The concerned two-phase flow is an object of mechanical cleaning before the processed air to be released to the environment. The first step of gas scrubbing is accomplished in a cyclone. The process of cleaning is because of presence of centrifugal forces in cyclone making solid particles to sediment and trap at the bottom of cyclone. The efficiency of the cyclone is defined as the ratio of the solid particles left the cyclone to the total number of particles entering cyclone. Size of the solid particles and flow rate are the parameters mainly influencing the cyclone efficiency.

Experimental study of the cyclone efficiency is a time consuming process which is one of the main reasons to study the flow behavior and cyclone efficiency numerically [1, 2, 3, 4, 5].

GEOMETRIC MODEL OF THE CYCLONE

Type and dimensions of the selected cyclone are presented in Figure 1. The presented cyclone is a middle-size used in industry.

The most important parameter of cyclone is its efficiency. It can be determined either experimentally or using CFD technology. Because the experimental study is a time consuming process the focus of the current study is to show if the CFD study is a reliable tool to evaluate the efficiency of the cyclone.

In a relation with the above a geometrical 3D model of the cyclone has been created (fig. 2). The generated volume includes in about 860 000 cells. On the figure 2 are pointed the elements of the cyclone.

The following initial conditions have been applied during the numerical procedure:

- Flue gases temperature at the inlet (3^{rd} stage of flue gases cooling) - 75 [^oC].
- Velocity at the inlet V_{gas} – 4, 6, 10 and 15 [m/s];
- Dust density ρ_p - 2100 kg/m^3;
- Mass flow rate of two-phase flow – [0,002 kg/s];

[141] Presented a parallel sessions room 9.3 report of October 29, 2016 with the original title: ЧИСЛЕНО МОДЕЛИРАНЕ НА ДВУФАЗНО ТЕЧЕНИЕ В ЦИКЛОН. АНАЛИЗ НА ЕФЕКТИВНОСТТА

- Minimum diameter of solid particles - 1 [μm];
- Maximum diameter of solid particles - 300 [μm];
- Average distribution - 130 [μm].

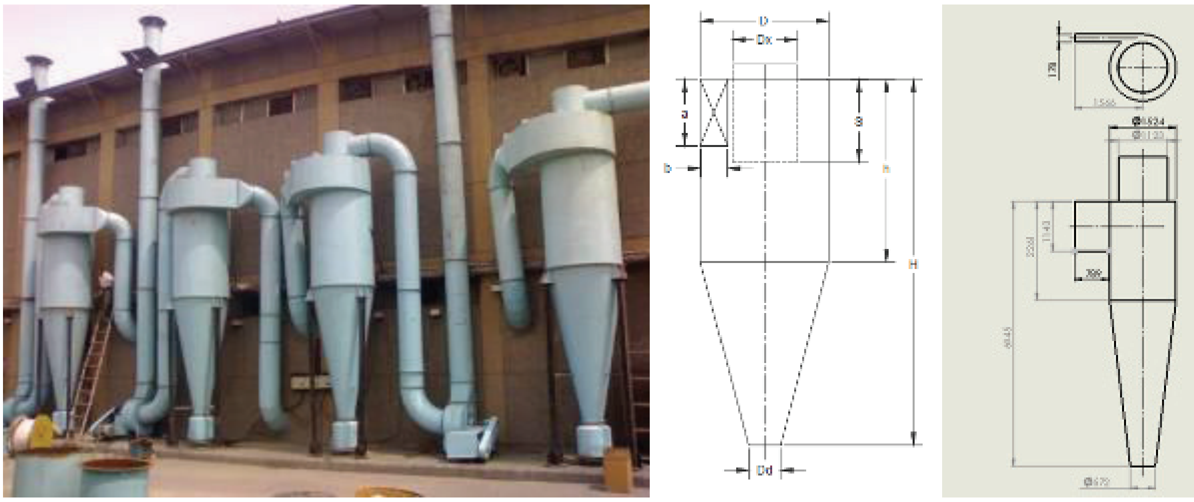

Fig. 1. Type and dimensions of selected cyclone

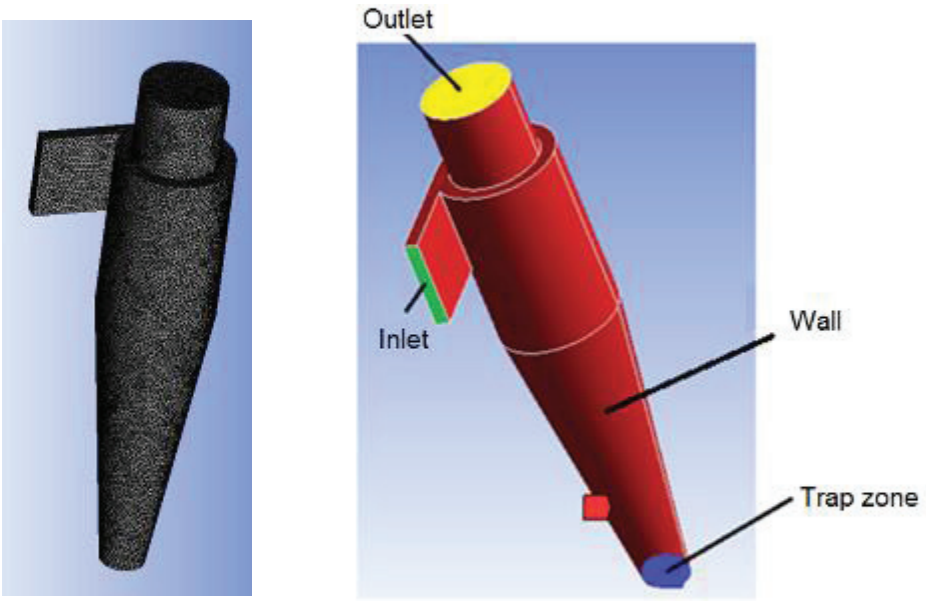

Fig. 2. 3D model of the cyclone

CFD STUDY. RESULTS

The developed two-phase flow is a swirl flow. The degree of the swirl flow depend both on the geometrical parameters of the cyclone and inlet parameters of the flow. The flow is turbulent with a prevailing rotational component of the flow. The RNG second order k-ε model of turbulence has been applied during the numerical procedure.

"Discrete phase model" was accepted. The total mass flow rate is 0,002 kg/s. Non-slip conditions were applied for the surface of the cyclone. A "Trap" Boundary condition for the bottom of cyclone was accepted.

4 different sections of the cyclone have been selected (Fig. 3) as below are presented the results of those sections.

Numerical procedure was performed at different initial velocities of the two-phase flow. The results are presented below.

Numerical procedure was performed at different initial velocities of the two-phase flow. The results are presented below.

Different initial velocities of the cyclone do not affect the outlet velocity field distribution (fig. 4). The velocity profile is very similar particularly with regard to peripheral velocity distribution.

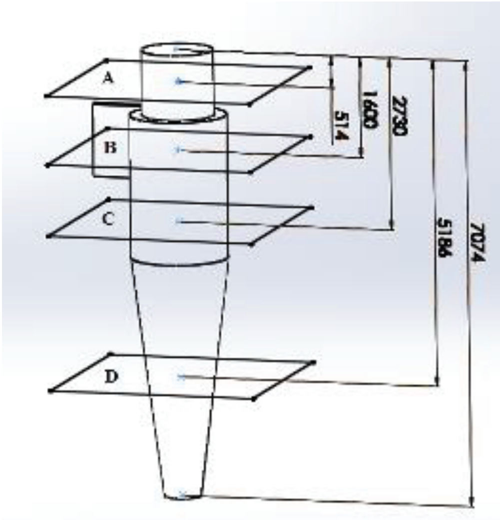

Fig. 3. Selected area cross section for the cyclone

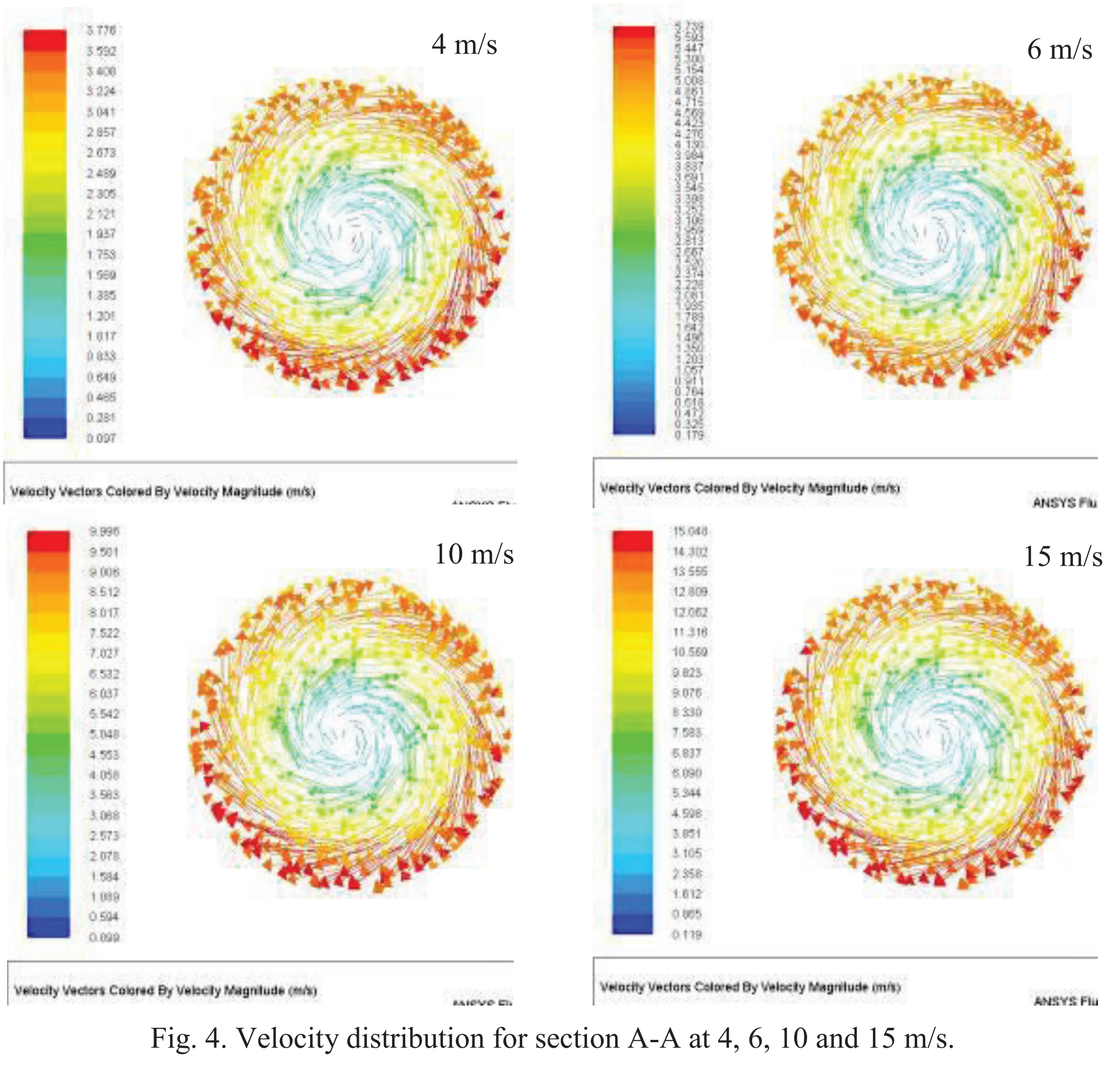

Fig. 4. Velocity distribution for section A-A at 4, 6, 10 and 15 m/s.

 Copyrights© 2016 ISSN 1311-3321

On fig. 5 are presented the velocity distribution for the other three sections.

The highest velocity can be observed close to the conjunction section between cylindrical and cone part of the cyclone.

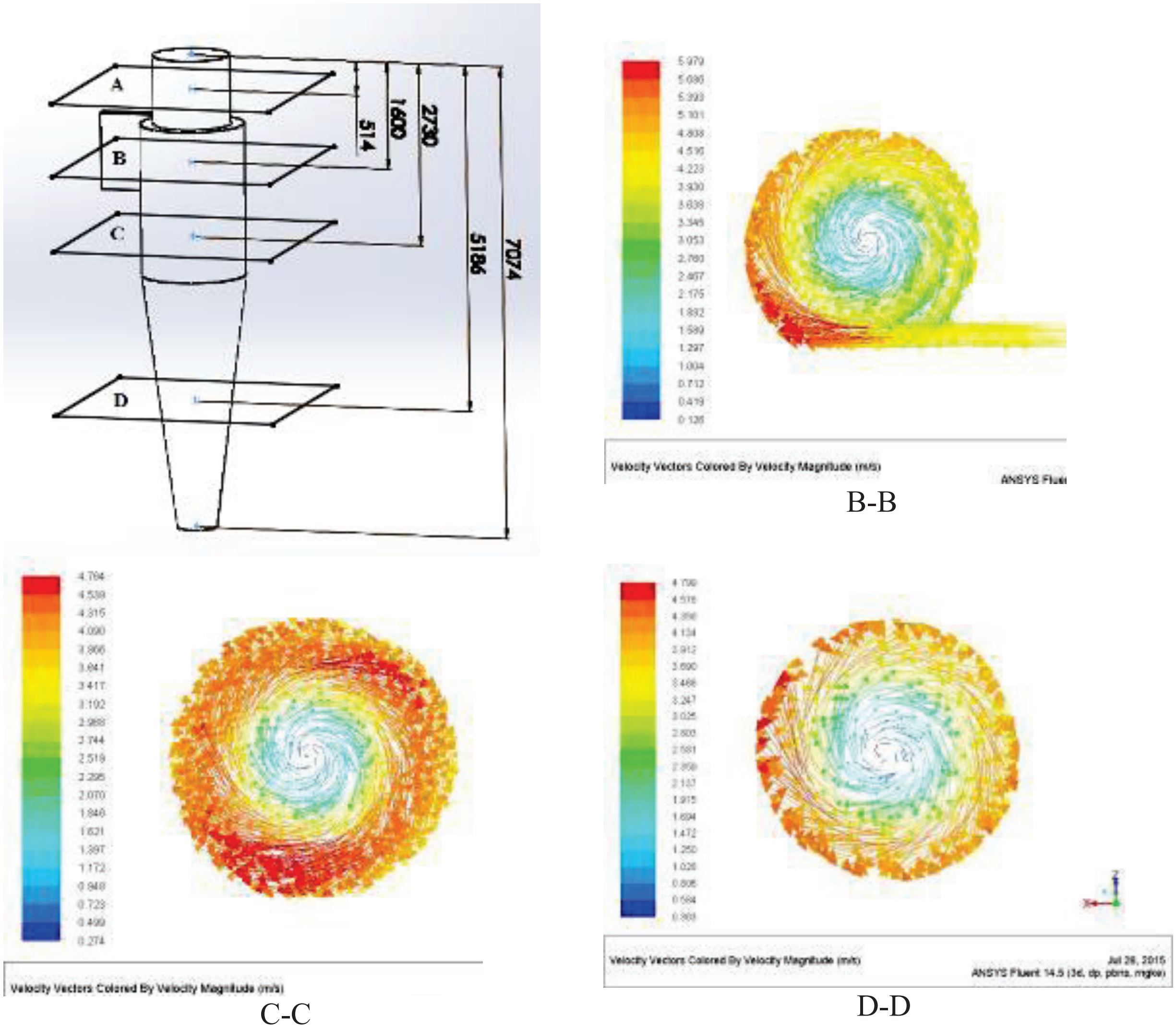

Fig. 5. Velocity distribution for sections B, C and D at 4 m/s initial velocity 6,5 m/s

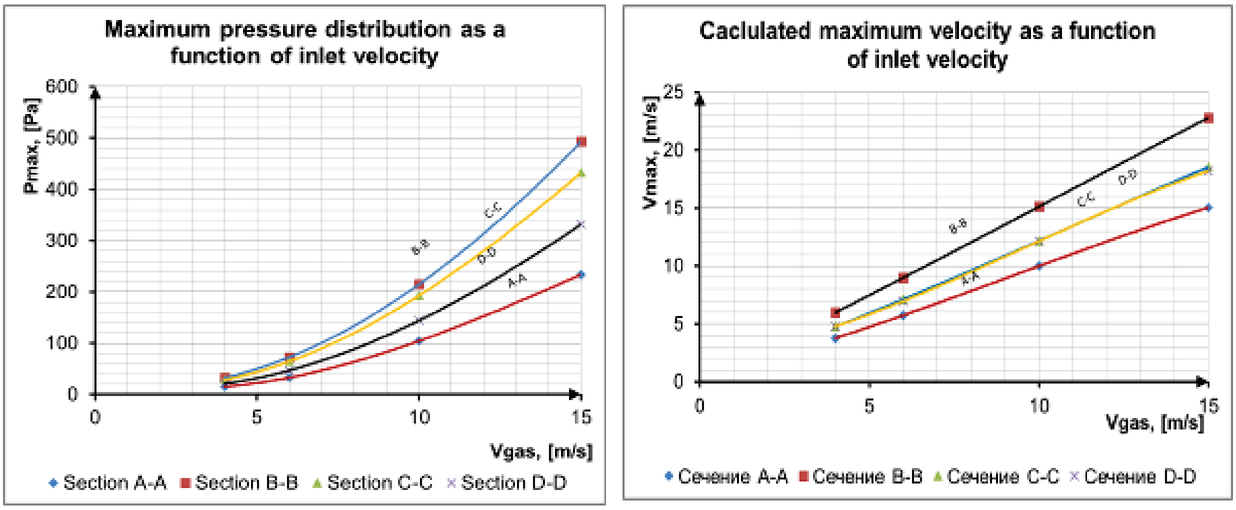

Fig. 6. Pressure and maximum velocity distribution as a function of inlet velocity

On fig. 6 is presented information about pressure and maximum velocity distribution in relation with inlet velocity of the cyclone. The lower pressure is at section “A” since the maximum pressure is achieved at section “B”. The same conclusion can be made for the maximum velocity for the selected cross sections.

On fig. 7 is presented the cyclone efficiency at different initial velocities. The figure shows that if the velocity at the inlet increases 3 times the efficiency decreases with 3%. Further increase of the velocity goes to the significant decrease of the efficiency of the cyclone.

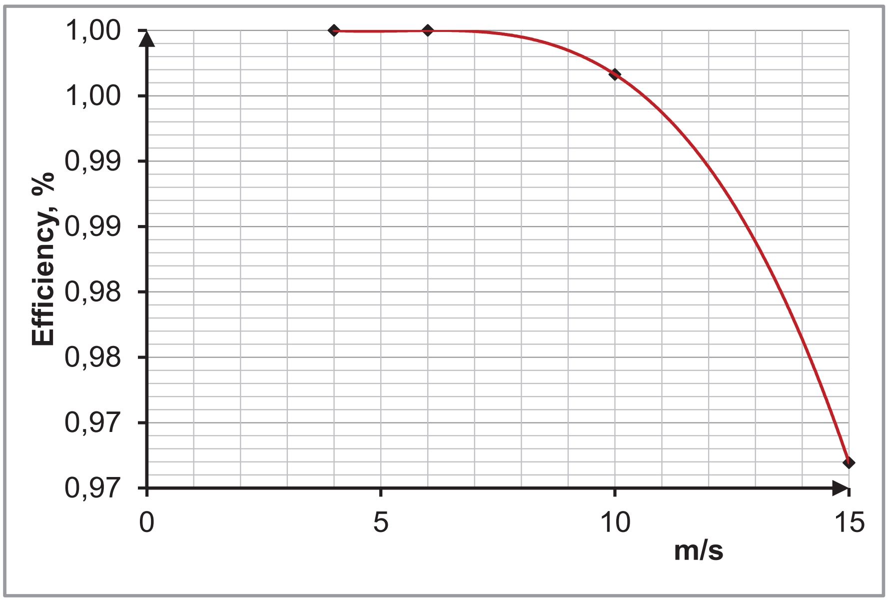

Fig. 7. Cyclone efficiency at different initial velocities

CONCLUSION

The current study shows that the applied NRG k-e model can be successfully used for the analysis of cyclone efficiency. The calculation time for CFD studies is reasonable. With the numerical study can be determined the velocity range for the cyclone operation for the desired level of gas cleaning. With the results obtained can be made suggestions for cyclones geometry improvements.

REFERENCE

[1] Kuang S., Z. Qi, et all, CFD modeling and analysis of the multiphase flow and performance of dense medium cyclones, Journal of Minerals Engineering, November, 2013;

[2] Chu K., J. Chen, A. Yu, Applicability of a coarse-grained CFD-DEM model on dense medium cyclone, Journal of Minerals Engineering, January, 2016;

[3] Safikhani H., Modeling and multi-objective Pareto optimization of new cyclone separators using CFD, ANNs and NSGA II algorithm, Journal of Advance Poweder Technolgy, September, 2016;

[4] Kepa A., The efficiency improvement of a large-diameter cyclone – The CFD calculations, Journal of Separation and Purification Technology, July, 2013;

[5] Song J., Y. Wei, G. Sun, J. Chen, Experimental and CFD study of particle deposition on outer syrface of vortex finder of a cyclone separator, Journal of Chemical Engineering, 2016

