

# Large Eddy Simulation of reverse-flow centrifugal separators

Gleb I. Pisarev\*, Alex C. Hoffmann\*, W. Peng\*,<sup>†</sup> and Henk A. Dijkstra\*\*

\**University of Bergen, Department of Physics and Technology, Allegt55, 5007 Bergen, Norway*

<sup>†</sup>*Aker Solution*

\*\**University of Utrecht, Department of Physics*

**Abstract.** Different CFD models of reverse-flow centrifugal separators namely swirl tubes have been built in order to study and detail analysis of the phenomenon of the "end of the vortex". The present work based on previous experience and experimental activities in the field of this phenomenon. The models was built in absolutely agree with geometrical characteristics and operating conditions of experimental works [1, 2]. Two different types of swirl tubes were analyzed. One in principle long tube with varied length was study to visualize behavior of the vortex core depending to the total length of separator. Other was built to improve the influence of so-called collection vessel and its depth on the phenomenon of "vortex end". Simulations were carried out using commercial CFD package Star-CD. Obtained results are qualitatively agreed with earlier obtained experimental data [2].

**Keywords:** cyclones, computational fluid dynamics, swirl tubes, end of the vortex, vortex length, turbulence, LES

## INTRODUCTION

Reverse-flow centrifugal separators are widely used for dedusting and demisting in industry. But in spite of general application there are some effects, which can occur during the process of separation using centrifugal gas cleaning equipment. One of them is the fluid flow phenomenon known as the "end of the vortex", which spontaneously occurs low in, or under, the separator. The point is that when gas call into the separator, at some conditions the vortex core deviates from the axis of the separator and attaches to the wall, where it rotates at some level from the bottom of the separator [2, 3]. The position and behavior of this end is crucial to proper separator performance, it influences efficiency, clogging and wear of cyclones operating individually or as part of a multiclone installation [4]. Computational fluid dynamics (CFD) study was carried out for a detailed understanding of reasons and conditions of this effect. The present work is focused on swirl tubes, which are often used in final stage of the gas cleaning process as high-efficiency dedusters operating at moderate or low-solids loadings. However in this work we are interested in flow pattern formed in the swirl tube; and conditions, under which the effect of the "vortex end" becomes strong, influential and visible. So, the simulations were carried out with pure air without particles and admixtures. There are many conditions that origin of the behavior of flow pattern. Owing to previous work we know that the total length of swirl tube, presence or absence of dustbin, cyclone wall roughness, diameter of gas outlet (vortex finder), geometry of dustbin, and inlet gas velocity are the main parameters of affecting to the flow behavior. Due to this, several CFD models of swirl tube with different geometry was built and investigated at some operating conditions.

## MODEL DESCRIPTION

The CFD models were built based on data of experimental work by Peng et al. [2]. One difference from real swirl tube consists in fact, that swirling of gas at the inlet was given by setting necessary value of components of inlet velocity in cylindrical coordinate system. In other worlds, the swirling vanes which force gas to rotate at the inlet of separator were not simulated. All other geometry and parameters of the process were keeping. In present work numerical grids consist of about 50000 cells and varied from one to other. In the case of swirl tube without dust finder (dustbin) number of cells was 39024, 54864, 70704 and 127728 for the total length of the separator 50, 70, 90, 162 cm respectively. In cases with dustbin cells number was 79344, 59184, and 49104 depending on the depth of dustbin. Mesh independence tests were carried out, to make sure that the solutions of CFD equations are not mesh dependent. As have been mentioned the working fluid was pure air with constant density of  $1.205 \text{ kg/m}^3$ . Diameter of separator and dustbin were 11 cm, and 28 cm respectively. The angle of gas coming in swirl tube was  $45^\circ$  in all cases. Other geometry and operating parameters were varied during investigations. The main point for numerical simulations was correct choice of model

of the turbulence to more probable description of the flow pattern in separator. There are many available turbulence models. But for instance standard k-ε model can not describe phenomenon that we interested in. Simulations using this model were carried out, but effect of deviation of the vortex core from vertical axis was not obtained at any conditions. Then LES turbulence model have been chosen. Large Eddy Simulation involves a three-dimensional, time-dependent computation of the large-scale turbulent motions responsible for turbulent mixing whilst those with scales smaller than the computational grid are modeled. The main difference between conventional turbulence modeling approaches and LES is the "averaging" procedure used to derive the equation of motion. The LES technique does not involve the use of ensemble averages, rather it consist in applying a spatial filter to the Navier-Stokes equations, which are recast in the following form [5]:

$$\frac{\partial}{\partial t} \bar{\rho} \langle u_i \rangle + \frac{\partial}{\partial x_j} \bar{\rho} \langle u_i u_j \rangle = - \frac{\partial \bar{p}}{\partial x_i} + \frac{\partial \bar{\tau}_{ij}}{\partial x_j} \quad (1)$$

In the above equation the filtered convection term needs to be modeled. This is achieved by introducing the sub-grid scale stress, defined as

$$\tau_{SGS,ij} = \bar{\rho} (\langle u_i u_j \rangle - \langle u_i \rangle \langle u_j \rangle) \quad (2)$$

In present study the Smagorinsky model has been chosen as a simplest, commonly used and computationally inexpensive type of LES. The main assumption of this model has the following form [6]:

$$\tau_{SGS,ij} - \frac{1}{3} \tau_{SGS,kk} \delta_{ij} = \underbrace{2 \bar{\rho} C_{s2} \Delta^2}_{\mu_t} \langle s_{ij} \rangle \quad (3)$$

where  $\langle s_{ij} \rangle$  represents the Frobenius norm of the strain rate tensor, and  $\Delta$  is taken as  $V_{cell}^{1/3}$ . The parameter  $C_{s2}$  is taken to be the square of the classic Smagorinsky constant (0.165) As recommended for LES turbulence the SIMPLE solution algorithm and Three Time Level Implicit temporal discretization have been chosen. Inlet velocity and gas exit were set as boundary conditions. No slip boundary condition was used in wall boundary; near wall treatment and roughness of the wall were standard wall functions. All of this equations, relations and assumptions are realized in the commercial flow code STAR-CD which has been used in present study.

## RESULTS AND DISCUSSION

The process of gas propagation in swirl tube was simulated.

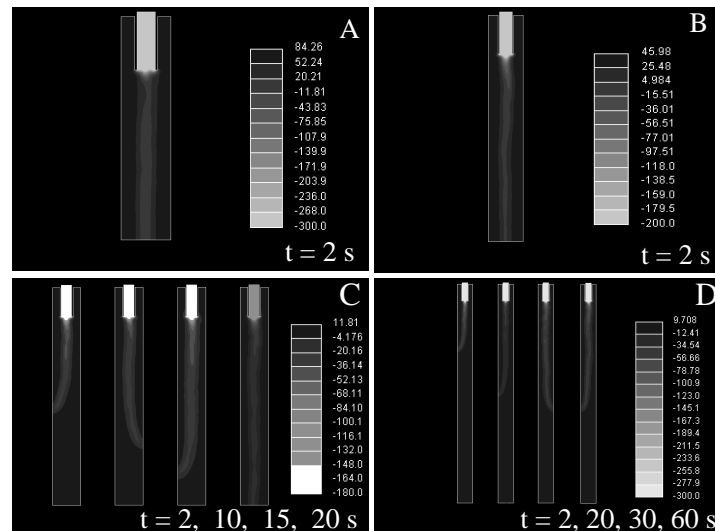


FIGURE 1. Contour plots of static pressure for different length of separator and time steps

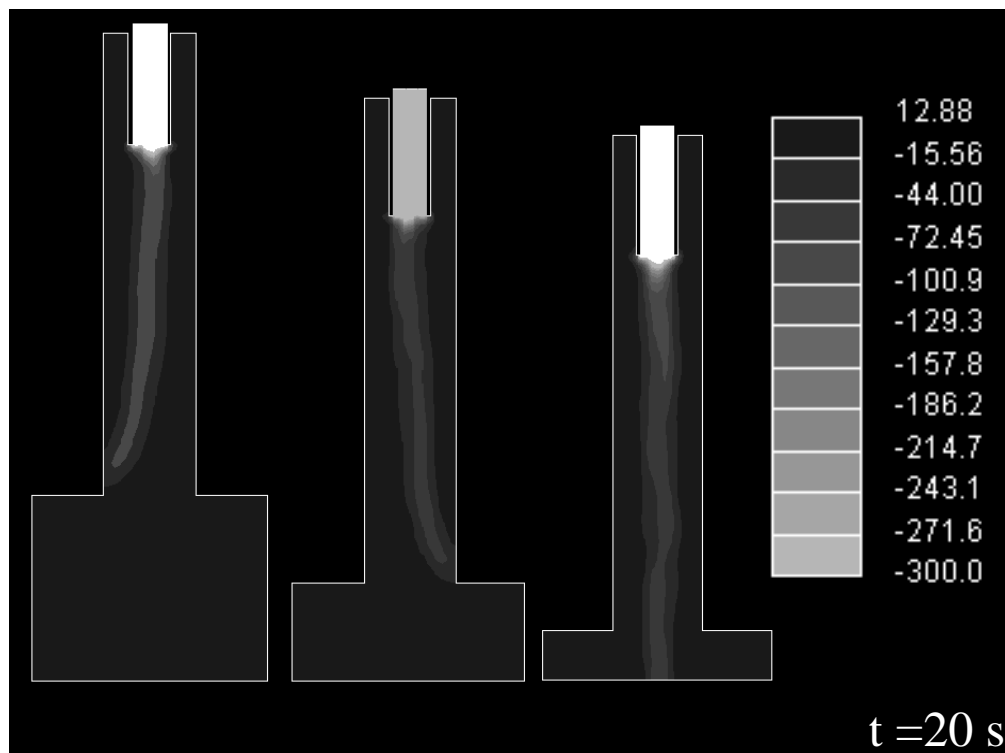
It was discovered that when the vortex is bending to the wall of separator and starts rotating on it, the level of rotating is not constant in the course of time. Parallel with rotating, in some cases the vortex core slightly goes down along the body of separator during some time. There are three options of behavior of vortex.

In the first one the vortex core centralizes (goes to the bottom of separator) during a very short period of time (about 1 sec.) (Figure 1A, Figure 1B). In other case, at the beginning of the process the core bends to the wall, starts rotating and goes down during some period of time (about 20 sec, this time depends on the inlet velocity) and then it reaches the bottom of separator (Figure 1C). At the last case the vortex core bends to the wall, starts rotating and goes down, but then it stops at some level from the bottom and starts rotating on it (Figure 1D).

Figure 1 lists all of these types of behavior at the same inlet conditions. Total length of the tubes was 50, 70, 90, 162 cm, respectively. Other geometrical characteristics were the same in all cases. Figure 1 shows the pressure distribution at any duration.

At high inlet velocities we obtained the same picture, but the time, during which the vortex goes to the bottom in the case when the length of the separator was equal to 90 cm, was shorter (about 13 sec.), and at the last case the vortex core stops at the level, much closer to the bottom as shown on Figure 1D.

The other important characteristic, which affects the phenomenon of "vortex end" is the presence or absence of so-called dust finder and its depth. The process of gas propagation was studied for three models of swirl tubes with a dust finder depth of 20, 10, 5 cm. The length of the part of separator without the dustbin was 50 cm for both cases. All other geometrical dimensions and boundary conditions were the same as in previous cases. Figure 2 shows the pressure at 20 sec of the process. Only in the last case, with a dustbin depth of 5 cm, we find the centralized vortex.



**FIGURE 2.** Contour plots of static pressure for different depth of collector vessel

In other cases we obtain, that at any inlet velocity the vortex core bends to the wall, quickly goes to the border of dustbin, and then it stops and rotates there during all the time. This phenomenon is qualitatively agreed with experimental data [2].

## CONCLUSIONS

Large Eddy simulations of reverse-flow centrifugal separator were carried out. CFD models of swirl tubes with different geometry were built. Numerically improved that flow pattern in such equipment has unstable nature and there are some type of behavior of the vortex core during the work. Depending of length of the separator and depth of the dustbin three possible pictures are realized:

- centralized vortex at any moment of time
- vortex core bend to the wall of the separator, start rotate on it and slowly goes down to the bottom and finally reach it after some period of time
- vortex core bend to the wall of the separator, start rotate on it, slowly goes down along the body of separator and stops at some level from the bottom

Obtained CFD results are qualitatively agreed with earlier experimental data.

## ACKNOWLEDGMENTS

Norwegian Research Council is gratefully acknowledged for funding of present work.

## REFERENCES

1. A. C. Hoffmann and L. E. Stein. Gas Cyclones and Swirl Tubes: Principles, Design and Operation, Springer Verlag, Berlin, Heidelberg, New York, 2002.
2. W. Peng, A. C. Hoffmann, H. W. A. Regelink and L. E. Stein. Experimental study of the vortex end in centrifugal separators: The nature of the vortex end, Chem. Eng. Sci., 60:6919-6928, 2005.
3. W. Peng, A. C. Hoffmann, P. Boot, A. Udding, H. W. A. Dries, A. Ekker, J. Kater. Flowpattern in reverse-flow centrifugal separators, Powder Technol. 127, 212-222, 2002.
4. Hoffmann, A. C., de Jonge, R., Arends, H. and Hanrats, C.. Evidence of the "Natural Vortex Length" and its Effect on the separation Efficiency of Gas Cyclones, Filtration and Separation. 32: 799-804, 1995.
5. John D. Anderson, JR. Computational Fluid Dynamics The Basics with Applications, International Editions 1995
6. J. H. Ferziger and M. Peric', Computational Methods for Fluid Dynamics, 3rd rev. Edition. Softcover, 2002