ANNALS OF THE ORADEA UNIVERSITY
Fascicle of Management and Technological Engineering
ISSUE #1, MAY 2014, http://www.imtuoradea.ro/auo.fmte/

CFD STUDY ON ELIMINATING IMPURITIES FROM NATURAL GAS FLOWS IN CYCLONE-TYPE SEPARATORS

Alina Maria GLIGOR[1], Valentin OLEKSIK[2], Cristian DEAC[3], Valentin PETRESCU[4]
[1] "Lucian Blaga" University of Sibiu, Romania, alina.gligor@ulbsibiu.ro
[2] "Lucian Blaga" University of Sibiu, Romania, valentin.oleksik@ulbsibiu.ro
[3] "Lucian Blaga" University of Sibiu, Romania, cristian.deac@ulbsibiu.ro
[4] "Lucian Blaga" University of Sibiu, Romania, valentin.petrescu@ulbsibiu.ro

Abstract**—**The quality of the natural gas extracted from deposits and transported through pipelines is a major factor in achieving the desired heating properties of the gas. A big problem related to the quality is the presence of impurities such as dust, sand or water. These can be removed by using separators. This paper presents an analysis of the efficiency of a cyclone-type separator in terms of speeds and pressure drops, using the computational fluid dynamics (CFD) method in the case of separating water droplets from the main gas flow.

Keywords**—**impurities; cyclone-type separator; speed and pressure profile

I. Introduction

THE safety of the functioning of the whole national natural gas transportation system depends foremost on the quality of natural gases, on the construction, exploitation and maintenance of the transportation system, but also on the purity of the natural gas throughout its path from extraction to the final user.

The commercialised natural gas must have specific characteristics and technological parameters that would allow their usage, such as a certain chemical composition, a high heat capacity, low contents of impurities and a guaranteed dew point.

The components contained in the natural gas or carried by it (e.g. water or solid particles) must be eliminated in order to satisfy the user's requirements, the valorisation of certain components (He, N_2, N_2S, C2÷C5+ etc.) and to avoid polluting the environment.

At a natural gas extracting well, the most important function of surface installations is to separate and retain liquid and solid impurities, brought up from the gas-bearing layer with the gas or resulting from condensation due to the decrease of the temperature on the extraction pipes.[1]

Solid impurities in natural gas can lead to an increased corrosion of the pipes and the technological installations, the premature wear of the piston-cylinder assembly and of the rotors of gas compressors, the hampering of technological processes at the consumers, their processing within various chemical processes, the decrease of the transportation efficiency and the increase of the technological risk related to transportation through natural gas pipelines.

The efficiency of the separation process depends on the type and dimensions of the separator, on the structure of the separator, on the speed of the gas current, on the physical properties of the gases and on the technological functioning regime (flow, pressure, temperature).

Gas separators can be divided into three categories [1]:

1) *Gravitational separators;*
2) *Centrifugal separators;*
3) *Mixed separators.*

In turn, centrifugal separators can be of two types:

1) *Rotational centrifugal separators, which use a helix device or a serpentine.*
2) *Inertial centrifugal separators - cyclones or multicyclones.*

In the current paper, the authors have realised an analysis of the efficiency of a cyclone-type separator, using the computational fluid dynamics technique.

II. The cyclone-type separator

Cyclone-type separators are based on the employment of the centrifugal force for separating liquid and solid particles from gases. Since the centrifugal force in a cyclone is increased significantly due to the small rotation radius and the high tangential speed, the separation efficiency is also higher than in other separators.

The separation is based on the phenomenon of sedimentation in centrifugal field, i.e. on the separation of liquids and solids in suspension based on the density difference. Gases are introduced in the cyclone sideways, through the helix channel, with a tangential speed of up to 11 m/s. The centrifugal force determines the pushing of the solid and liquid particles towards the inner mantle. The helix shape of the centrifugation channel leads to the gas being given also a descending speed [1].

After centrifugation, the gases enter the main chamber of the separator, where the ascending speed is much lower than the descending speed in the centrifugation channel. This causes the solid and liquid particles to drop without problems into a high pressure reservoir, where the water and the solid impurities will be separated. The impurities can be evacuated manually or using automation devices [1], [2].

Cyclones are usually employed at the entrance in natural gas regulation and measurement units or in compression units.

The centrifugal force acting on the particles, F_c, can be determined using the formula:

$$F_c = k_c \frac{m_p \cdot v^2}{R} \qquad (1)$$

where:
m_p is the particle's mass, in kg;
v is the particle's speed, in m/s;
R is the radius of the cyclone, in m;
k_c is a proportionality coefficient taking into account the difference between the specific mass of the particles and that of the air, but also the particles' size.

The efficiency of a cyclone, β, is calculated with the formula:

$$= \frac{\rho_p \cdot d_p (R_e - R_i)}{\mu \cdot R_e} \cdot v \cdot k_\Psi \qquad (2)$$

where:
R_e and R_i are the cyclone's outer and inner radiuses, respectively, in m;
k_Ψ is a shape factor taking into account the distribution of the particles' diameters.

Cyclones can be characterised by a large array of functional parameters:

1) *speeds of up to 50 m/s, depending however on the speed of the gases through the pipe at low pressure, needed for keeping the pressure drop within reasonable limits;*
2) *various separation depths, up to completely eliminating the particles with a size of 5µm and 80…85% of the particles with a size of 3µm;*
3) *pressure drops of 10...12 mbar for simple, low-pressure cyclones or of around 60 mbar for cyclones situated on the transport pipeline.*

In high performance cyclones there can be noticed following aspects:

1) *the separators's body consists of two cylindrical vessels joined through flanges, the lower part having a larger diameter and serving as impurities collection basin;*
2) *at the end of each helix-shaped part of the cyclone, there is welded a metal plate for separating from the gas flow the liquid drops and the solid particles that were transported towards the exit.*

III. COMPUTATIONAL FLUID DYNAMICS

In the last few years, the numerical simulation methods have become more and more present in applications related to simulating the behavior of fluids. Within the domain of fluids flow, the corresponding study method is called Computational Fluid Dynamics (CFD) and comprises the methods for calculating and analysing systems that involve fluids flow, heat transfer and associated phenomena such as chemical reactions, using computer-aided simulations.

Using the CFD method for a better understanding of the fluids flow processes represents an important step because it offers information that cannot be obtained otherwise. The CFD method is more and more used in modelling systems that include fluids flow, in many areas. The CFD codes enable the numerical solving of fluids transportation and of mass and energy balances in systems with a very complicated geometry. The results are specific models of the flow, that are very difficult to obtain experimentally or by using conventional modelling methods.

In order to achieve satisfactory results, it is extremely important to select an adequate turbulence model or a suitable numerical solving scheme [3].

The outstanding increase of the computing power in the last years has led to CFD becoming one of the domains with the highest increase in research.

By using the CFD simulation technique and the unstructured grid model for the cyclone geometry, it becomes possible to determine and describe in detail the flowing and separation processes for a two-phase solid-liquid mixture in the cyclone, which can then be used for creating even more precise models.

The commercial CFD software packages use for solving the fundamental partial differential equations for the conservation of the kinetic moment or of the scalars such as mass, energy or turbulence, a form of discrete integration, based on the volume control technique, which comprises three main steps [4]-[5]:

1) *Dividing the fluid domain into discrete control volumes by using a mesh of calculus nodes. The discretization involves substituting a variable form the integral flow equations (representing convection processes, diffusion processes etc.); these transform the integral equations into an algebraic equation system.*
2) *Integrating the fundamental equations on the control volumes in order to determine the unknowns (pressure, speed or other scalars).*
3) *Solving the discretized algebraic equations by means of an iterative method.*

IV. RESULTS AND DISCUSSIONS

A. The speeds profile in the cyclone

The authors have chosen to study the case when the cyclone-type separator is used to separate water droplets out of a current of natural gas.

For this, a gas-water mixture of 80% gas and 20% water droplets has been considered to enter the cyclone-type separator and using the computational fluid dynamics technique it has been intended to study the distribution of speeds and then of pressures in the cyclone.

In order to study the distribution of speeds in the cyclone, there have been considered axial and transversal cross-sections through it, for which the graphical representations of the speed vectors were then generated. The speed profiles were studied also near the cyclone wall.

As was expected, the flow was channeled near or inside the cyclone, due to the presence of flow constraints imposed to the fluid domain. Also it is easy to distinguish a strong radial flow from the central axis of the cyclone towards its walls.

The distribution of the tangential speed in the cyclone is influenced mainly by the functioning parameters, by the admission and rotation speed and by some geometrical parameters. Fig.1 shows the behavior of the gas-impurities mixture from the point of view of the speed variation, while Fig. 2 and Fig. 3 present the speed variation for the pure gas particles and for the water particles, respectively.

In the separation area, the rotation speed has a strong effect on the distribution of the tangential speed. Since several studies [6]-[8] associate the tangential speed with the cyclone's separation efficiency, an increased tangential speed indicates a better separation efficiency.

B. The pressures profile in the cyclone

The pressure drop in the cyclone reflects the energy needed for a separation process and represents a direct measure of operating costs, so its assessment is very important in designing a cyclone. For a cyclone which has to separate mixtures with low concentration, the pressure drop can be divided into a dissipated pressure drop and an efficient pressure drop.

An increase in the separation ratio leads to a decrease of the efficient pressure, while the diameter of the overflow hole influences the efficient pressure drop in a reverse manner.

The pressure in the outer helix current is larger than the one in the inner helix current. For the same cross-section area, the pressure is larger at the cyclone wall and decreases, first slowly but then suddenly, in the vicinity of the cyclone's axis.

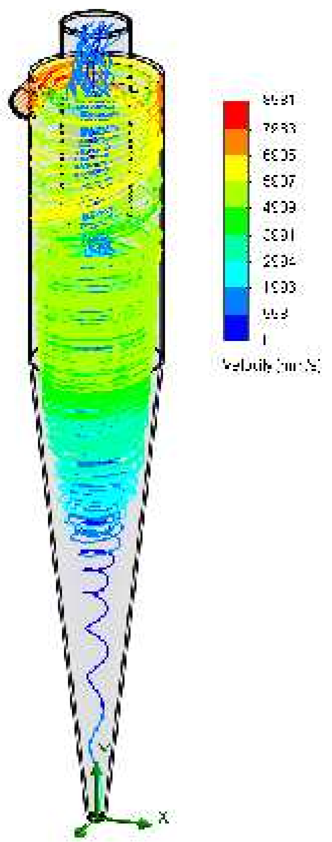

Fig. 1. Speed of the gas-impurities mixture inside the cyclone

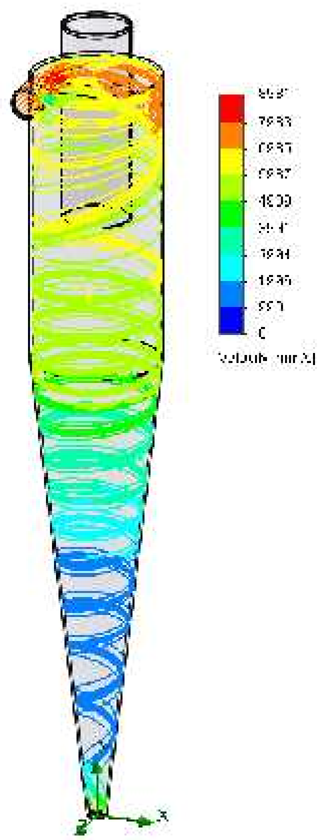

Fig. 2. Speed of the impurities inside the cyclone

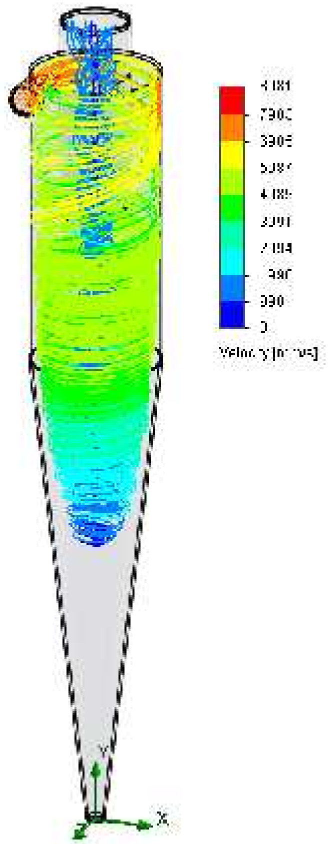

Fig. 3. Speed of the purified gas inside the cyclone

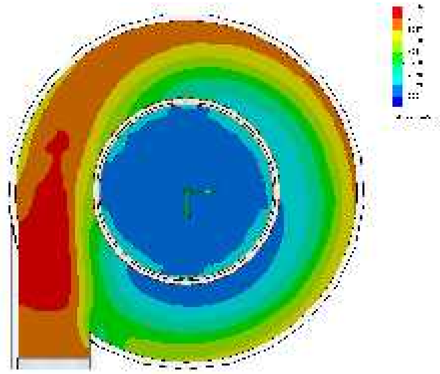

Fig. 4 Speed of the gas-impurities mixture in a cross-section located at the entrance into the cyclone

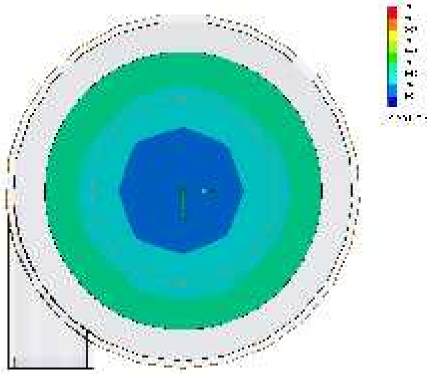

Fig. 5. Speed of the gas-impurities mixture in a cross-section located at 900 (mm) from the entrance into the cyclone

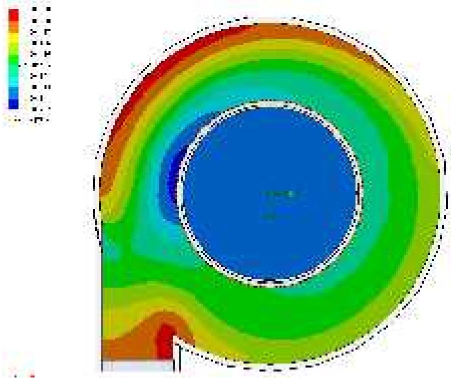

Fig.6 Pressure of the gas-impurities mixture in a cross-section located at the entrance into the cyclone

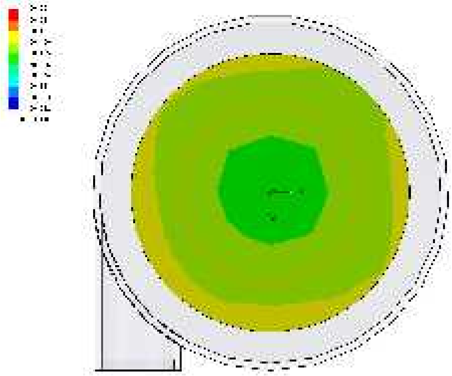

Fig. 7. Pressure of the gas-impurities mixture in a cross-section located at 900 (mm) from the entrance into the cyclone

V. CONCLUSIONS

On world level there is a tough competition in the realising of efficient centrifugal separators, for pushing the performances as high as possible and for reducing the exploitation costs as much as possible.

An adequate analysis of the flow inside a cyclone-type centrifugal separator can lead to an optimal design of the centrifugal separators and to ensuring a high efficiency in separating solid particles from a multiphase mixture.

Consequently, the application of numerical analysis for such applications is more than necessary and methods such as the computational fluid dynamics method can lead to a significant improvement in the understanding of phenomena inside the cyclone and thus to a more efficient design of separators.

In future, the authors intend to refine the model used for the computational fluid dynamics method, in order to obtain more accurate results that can lead to improved efficiency levels in the separation of impurities from natural gas flows.

REFERENCES

[1] N. Simescu, D. Chisalita, "Conditioning of natural gases (Conditionarea gazelor naturale)", Publishing House of the ”Lucian Blaga” University of Sibiu, 2001

[2] N.P. Cheremisinoff, "Handbook of Chemical Processing Equipment", Butterworth-Heinemann, 2000.

[3] J.G. Severino, "Mechanistic modeling of solid-liquid separation in small diameter hydrocyclones", Doctoral Thesis, University of Tulsa, USA, 2007

[4] G. Ipate, T. Casandroiu, "Computer simulation of liquid-solid separation - hydrocyclone with two conical sections", MOCM-13, vol.2, Romanian Technical Sciences Academy, 2007.

[5] G. Ipate, "Studies and researches on the separation process of suspensions for modernizing equipments from wastewater cleaning installations (Studii şi cercetări privind procesul de separare a suspensiilor în vederea modernizării echipamentelor din instalaţiile de epurare a apelor uzate)", Ph.D. thesis, Bucharest, 2011.

[6] B. Zhao, Y. Su, J. Zhang, "Simulation of gas flow pattern and separation efficiency in cyclone with conventional single and spiral double inlet configuration", Chemical Engineering Research and Design, A12(84), p. 1158-1165, 2006

[7] F. Qian, Z. Huang, G. Chen, M. Zhang, "Numerical study of the separation characteristics in a cyclone of different inlet particle concentrations", Computers and Chemical Engineering, 31, p. 1111-1122, 2006.

[8] N. Kharoua, L. Khezzar, Z. Nemouchi, "Study of the Pressure Drop and Flow Field in Standard Gas Cyclone Models Using the Granular Model", International Journal of Chemical Engineering Volume 2011, Article ID 791218, 2011