ACCEPTED MANUSCRIPT

Technical Sciences

Title: Analysis of a cylindrical cyclone separator used in aircraft turbine engine

Authors: Tomasz Szwarc, Włodzimierz Wróblewski, Tomasz Borzęcki

Analysis of a cylindrical cyclone separator used in aircraft turbine engine

Tomasz Szwarc[1,2], Włodzimierz Wróblewski[1], Tomasz Borzęcki[2]

[1] Silesian University of Technology, Department of Power Engineering and Turbomachinery, ul. Konarskiego 18, 44-100 Gliwice, Poland
[2] Avio Polska sp. z o. o., ul. Grażyńskiego 141,43-300 Bielsko-Biala, Poland

Abstract

Cyclone separators are components commonly used in the oil system of aircraft gas turbine engines to separate air from the oil. The major advantage of this component is its simple construction and high reliability thus it does not require frequent inspections. The efficiency of the separator has a decisive impact on the quality of the oil, which affects directly the efficiency of the oil system. New generations of engines require more compact separator designs to reduce weight and project costs while maintaining (and often increasing) engine's efficiency and reliability. To meet these requirements and optimize the construction of the separator, flow modelling of the air-oil mixture must be used in the design process. The purpose of this paper is to present a numerical simulation approach using the volume of fluid model for aircraft turbine separator.

Keywords: Multi-phase flows, air-oil separator, gas turbine, cyclone, volume of fluid method

1. Introduction

Cyclone separator is a component installed inside the main engine oil tank (Figure 1). The major advantage of this component is its simple construction thanks to which it does not require maintenance and frequent inspections. Its role is to separate air from the oil. The air-oil mixture is generated during the lubrication process inside the engine, bearing, sump and gearbox cavities. The efficiency of the separator has a decisive impact on oil quality, affecting directly the efficiency of the oil system. Increased air content in the oil causes a pressure drop in the system and higher-pressure fluctuations, what in turn affects proper lubrication and cooling of the engine components (bearings, gears, splines, accessories). Compared to typical industrial cyclone separators, those used in aircraft need to meet many more requirements. Constantly changing operating conditions during flight missions cause changes in many parameters like air/oil ratio, oil tank level, pressure,

attitude, position. High swirling flow inside the separator is frequent and not stable. Unstable distribution of the flow field determines the separation performance.

In this paper, a numerical model of air-oil separator under one operating condition is analyzed. The flow field in the separator is complex and may require including the swirling and anisotropy phenomena. Although some papers [2], [3], [6], [8], [12], [15] suggest using the Reynolds Stress Model (RSM) or Large Eddy Simulation (LES), these advanced models are not suitable for the optimization algorithms, as this will be the goal of future investigation. The above mentioned models are time-consuming and quite challenging in terms of the calculation hardware needed. In the industrial approach, two equation models are more commonly used [13]. In this study, the RNG k-ε model with swirling option is applied. The model can recreate the flow characteristics in a cyclone separator, which can be useful for designing and optimization of this engine component [14]. The impact of different turbulence models on the solution will be studied in the next step. For this preliminary analysis, turbulence was modelled using the two equation RNG k-ε model as it enables faster calculations [14] (2 equations compared to 7 in RSM) [14]. For the simulation of the two-phase flow, the volume of the fluid model was selected. It allows to include free-surface [9] in the oil tank in the analysis, the location of which also has an impact on the separator performance [10]. Although gas-liquid separators (e.g. [1], [5], [10]) and other applications in the aero-engine (e.g. [4], [7]) have been studied and discussed, there are no available papers describing cyclone used in an aircraft engine oil system. Typical cases reported in the literature deal with applications in the oil and gas industry, with different geometry or air/oil ratio, what makes the comparison hard. Other publications present cyclone separators with geometry similar to the one presented herein [3], [12] but they are used in the mineral industry with different fluid types or fluids mixed with solid particles.

The purpose of this paper is to present a preliminary numerical approach for the analyses of the two-phase fluid flow. The main goal is to prepare the numerical model which can be implemented in the optimization procedure. In further steps, the recommendations for air-oil separator designers will be formulated.
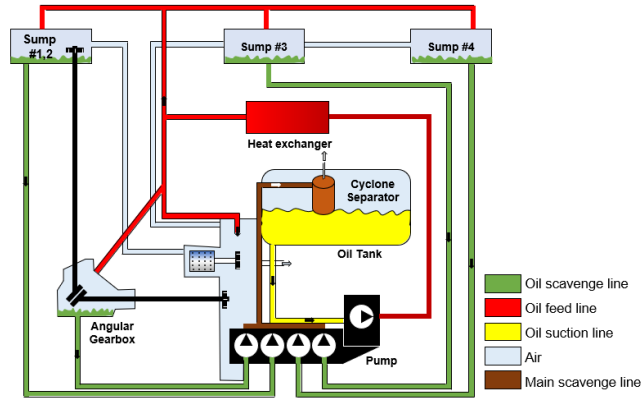
Figure 1. Simplified scheme of an aircraft oil system

In the gas turbine engine, oil is used to lubricate and cool components - bearings, gears, spline joints and dynamic seals. In the lubrication process, the oil is aerated in sumps and gearboxes. It flows by gravity to the lower parts of the engine where it is sucked in by the scavenge pump and directed to the oil tank. The efficiency of the oil separator has a decisive impact on oil quality. Deterioration of its characteristics causes reactions throughout the lubrication and cooling circuit. Considering the CFD analysis, a study of a two-phase model selection must be conducted. Selection of the appropriate mathematical model depends on the flow type at the separator inlet. This is crucial for the proper simulation of the processes in the separator. The first selection can be made based on two-phase flow maps [13].

2. Numerical model

2.1. Geometry and boundary conditions

The geometry of calculation domain was created based on the geometry of the existing test bench where the separator was installed in a cylindrical tank shown in Figure 2. The air-oil mixture flowed into the tank tangentially and a swirl was generated inside the separator. The oil was directed towards the walls and then flowed down into the tank. The separator has two outlets: one in the upper part and one in the bottom. The air was extracted from the separator through the upper outlet and the presumably clear oil flowed through the bottom outlet. At the inlet, the mass flow rates of both air and oil were applied as boundary conditions. The uniform velocity with normal direction to the inflow cross-section was assumed. The temperature of the mixture was constant. At the outlets, the pressure conditions were specified.

The experimental data were utilized to develop the simulation model. Test bench allows to measure the inlet mass flow rate of each fraction at the inlet and the outlet, pressure and temperature are measured only at selected points. Oil quality defined by Eq. (1) was measured at the outlet line by checking the volume fraction of each phase. Oil properties used in the simulation were for standard aviation oil used in gas turbine engines. At the inlet, the volumetric air/oil ratio in the experiment was equal to 1.3. Numerical schemes for this analysis were selected based on the analysis performed in [15]. As solution algorithms for pressure-velocity coupling, a coupled scheme was selected with pseudo transient option. For the preliminary numerical analysis, the first order upwind schemes were set for turbulent kinematic energy and dissipation rate.
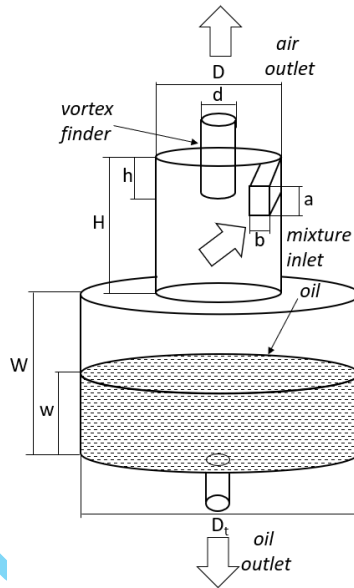

Figure 2. Separator with test tank

It was necessary to perform a study to identify the proper boundary conditions at the outlets, which would enable mapping the conditions prevailing at the measuring stand. The performance of the separation process, which is the object of this calculation, is described by two coefficients. One characterizes the amount of gas leaving through the oil outlet while the other indicates the amount of oil leaving through the air outlet. These values were measured during the experiment. The oil quality coefficient (OQ) is defined as the volume fraction of oil at the oil outlet

$$OQ = \left(\frac{\dot{V}_o}{\dot{V}_a + \dot{V}_o}\right)_{oil\ outlet} \tag{1}$$

Separation efficiency η_s is defined as the ratio of the difference between the volume rate of oil at the inlet and the volume flow rate at the vent to the volume rate of oil at the inlet:

$$\eta_s = \frac{\dot{V}_{o,inlet} - \dot{V}_{o,vent}}{\dot{V}_{o,inlet}} \tag{2}$$

The major challenge in defining the simulation were properly set boundary conditions. Since air and oil masses were measured and mixed before they got into the separator, the mass flow rate with a homogeneous distribution of the second phase was selected.

At both oil and air outlets – the pressure values were set. This set of boundary conditions allows to control the process in the separator. The other possible types of boundary conditions which are based on mass flow value could not be applied at the outlets since mass flow rates should be the results of computation analysis. The dimensions of the test bench are presented in Table 1.

Table 1. Dimensions of the cyclone with tangential inlet

Dimensions	Cyclone height (H)	Vortex finder height (h)	Cyclone diameter (D)	Vortex finder diameter (d)	Inlet tube height (a)	Inlet tube width (b)	Tank height (W)	Tank diam. (D$_t$)
Value in parameter (a)	3.2	1.4	3.6	1.3	1	1.6	12	3

2.2. Mathematical model of the two-phase flow

To simulate the separation phenomena in the air-oil mixture the Volume of Fluid (VoF) model is used. This model allows for tracking the air-oil interface. The VoF is used to model the flow of two immiscible fluids. The continuity equation is solved:

$$\frac{\partial}{\partial t}(\rho) + \nabla \cdot (\rho \vec{v}) = 0 \tag{3}$$

where $\vec{v}$ is the velocity vector, ρ is the mixture density calculated from:

$$\rho = \alpha_a \rho_a + (1 - \alpha_a)\rho_o \tag{4}$$

where ρ_a, ρ_o are densities of air and oil respectively and α_a is the volume fraction of the air which is, in this case, the second phase. The interface between the phases is tracked using the continuity equation for the volume fraction of air:

$$\frac{\partial}{\partial t}(\alpha_a \rho_a) + \nabla \cdot (\alpha_a \rho_a \vec{v}_a) = \dot{m}_{oa} - \dot{m}_{ao} \tag{5}$$

where $\dot{m}_{oa}$, $\dot{m}_{ao}$ are the mass flow rates from oil to air and air to oil respectively. The volume fraction equation is not solved for the oil phase (primary); the oil-phase volume fraction is computed based on the following constraint:

$$\alpha_a + \alpha_o = 1 \tag{6}$$

For this case, the volume fraction equation is solved through the implicit formula. A single momentum equation is solved throughout the domain, and the resulting velocity field is shared among the phases. The momentum equation, shown below, is dependent on the volume fractions of all phases through the mixture density ρ and mixture viscosity μ [11]:

$$\frac{\partial}{\partial t}(\rho \vec{v}) + \nabla \cdot (\rho \vec{v} \vec{v}) = -\nabla p + \nabla \cdot \left[\mu_{eff}(\nabla \vec{v} + \nabla \vec{v}^T) \right] + \rho \vec{g} + \vec{F} \tag{7}$$

where $\mu_{eff} = \mu + \mu_t$ is the dynamic effective viscosity, $\vec{g}$ is the gravitational acceleration, p is the pressure, $\vec{F}$ is the surface tension source term.

2.3. Turbulence

The RNG k–ε model is derived from the instantaneous Navier–Stokes equations using a mathematical technique known as the "renormalization group" (RNG) methods. It is based on the standard k-ε model but includes refinement. An additional term in the ε equation improves the accuracy for rapidly strained flows. By this, the effect of swirl on the turbulence is included, enhancing the accuracy for swirling flows. The RNG theory provides an analytical formula for the turbulent Prandtl numbers, while the standard k-ε model uses user-specified, constant values [16].

The equations for turbulence kinetic energy k and dissipation rate of turbulent kinetic energy are solved:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k U_i) = \frac{\partial}{\partial x_j}\left(\alpha_k \mu_t \frac{\partial k}{\partial x_j} \right) + G_k + G_b - \rho \varepsilon - Y_M + S_k \tag{8}$$

$$\frac{\partial}{\partial t}(\rho \varepsilon) + \frac{\partial}{\partial x_i}(\rho \varepsilon U_i) = \frac{\partial}{\partial x_j}\left(\alpha_\varepsilon \mu_t \frac{\partial \varepsilon}{\partial x_j} \right) + C_{1\varepsilon} \frac{\varepsilon}{k} G_k - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} - R_\varepsilon \tag{9}$$

In the above equations α_k and α_ε are the inversed effective Prandtl number for k and ε. $C_{1\varepsilon}$ and $C_{2\varepsilon}$ are constant values of 1.42 and 1.68. The scale elimination procedure in the RNG theory results in a differential equation for the turbulent viscosity. In the high-Reynolds number limit the effective viscosity μ_t is given by:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \tag{10}$$

with C_μ=0.0837 derived using the RNG theory. It is interesting to note that this value of C_μ= is very close to the empirically determined value of 0.09 used in the standard k-ε model [16].

2.4. Mesh

Separator geometry was divided into a tetra mesh which was generated using Ansys Workbench tool 19.2. The 3D geometry was meshed using different numbers of elements from 512 to 1527 thousand tetra cells. Mesh was refined mainly in the regions close to the wall, a moderate refinement was introduced in the central part of the separator domain. The quality of all 3D computational meshes with different numbers of elements has been checked before the simulation.

Table 2. Mesh quality

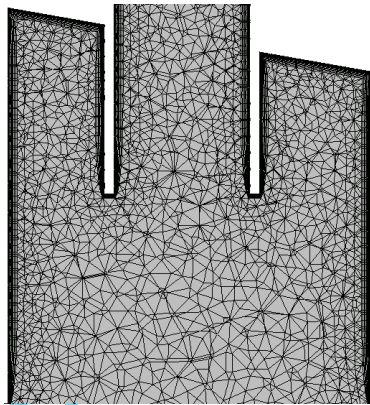| Key factor	Requirement	Mean mesh quality
Aspect ratio	5:1	5.34
Orthogonal quality	>0.01 (best cell closer to 1)	0.96
Skew	Below 0.95	0.82

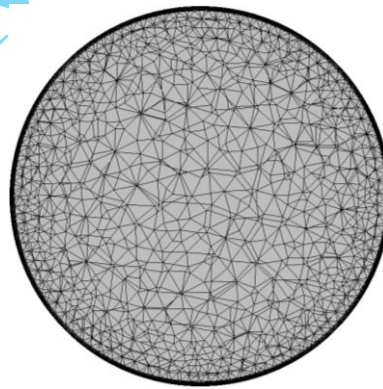**Figure 3.** Mesh at the cross-section of the separator

Figure 4. Mesh at the bottom outlet of the separator

After reviewing the results of the first coarse mesh, it was noticed that the model was giving the OQ value at a satisfactory level, but the stability of the solution was very poor. To understand the impact of the size of the numerical grid, the mesh independence study was performed. Two parameters were considered for the stability of the analysis: air and oil mass imbalance. The imbalance is defined as the sum of mass flows (inlet mass flow – outlets mass flows). For what

concerns the impact of the mesh on the parameters of interest, the amplitude of the air mass flow rate at the outlet, oil quality and oil volume in the domain were selected. Figure 3 presents a summary of this study. The increase of mesh density does not affect significantly the oil quality (maximum difference of 13% is observed). Also, the oil level inside the domain remained stable. The improvement was observed for parameters related to the stability of analysis. Considering OQ as the only goal of analysis, the mesh independence study reveals that changes of the mesh refinement were relatively small. The possibility of selecting mesh of lower density could be a significant advantage in the optimization process.
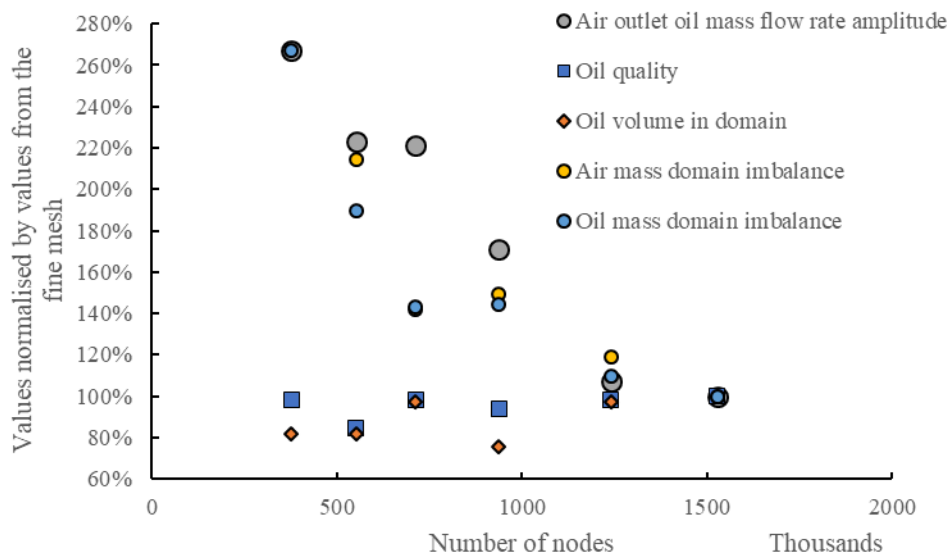

Figure 5. Normalized solution parameters vs the number of nodes

After selecting the appropriate type and density of the mesh, further goals are to optimize the geometry of the presented separator. Therefore, the currently selected mesh with 124k tetra cells can ensure the acceptable accuracy of results and a reasonable calculation time. This is important because the computation time of one case is about 1-2 weeks using a computer cluster.

3. Calculation results

This paper aims to present a numerical approach to cyclone separator analyses. The primary phase was set as the oil and the secondary phase was the air. The analysis process began to get complicated already at the initial condition formulation. The initialization was very important for calculation. Wrong setting of initial values causes convergence problems or solver crash. In case of the incorrectly set inlet pressure, a reverse flow was created at both outlets. It was impacting

the solver convergence and caused mass imbalance problems. Pressure value at the inlet was set based on the already known experimental value of the pressure drop across the separator. The preliminary simulation was carried out with a very coarse mesh to get an impression of how to implement working conditions. The analysis was initialized with the empty tank which was filled until the stabile oil level was reached. The initialization with a specific oil level caused solver crash.

The location of the free surface in the oil tank is very sensitive to boundary conditions. It is an important factor because it will directly affect the formation of the flow structures in the separator and, as a consequence, its efficiency as predicted in [10]. This is a characteristic feature of open separators, where the shape of the tank combined with the oil level can impact performance. The main flow swirl in the cylinder causes oil separation and the oil deposits on the separator wall. The main flow is two-phase. The first phase was parallel to the outlet of the separator and the second went down the cylinder, creating a helical shape of streamlines (Figure 7 and Figure 9). The first flow caused the oil concentration on the top corner of the separator, which is unlikely to happen due to recirculation with the incoming mixture. The second oil stream entering the tank influenced the oil free-surface.
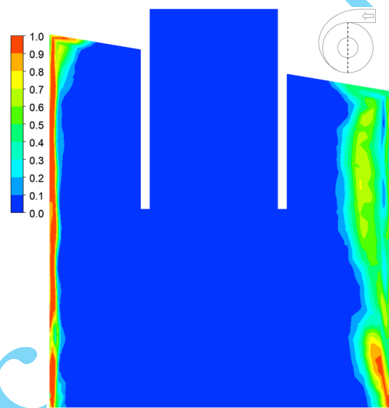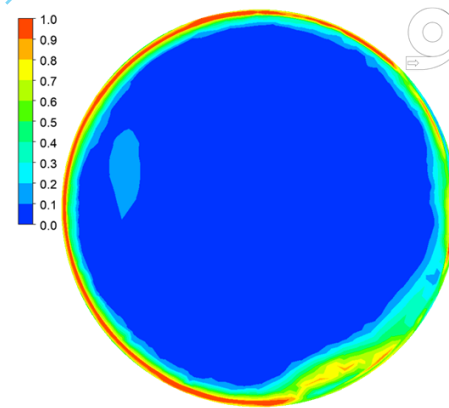

Figure 6. Oil volume faction contour

Figure 7. Oil volume faction contour (bottom outlet of the separator)

In Figure 6, formation of oil film on the left wall is visible. The geometry of the inlet to the separation zone has a key impact on separation phenomena since it can break the already formed film (Figure 7). The axis of the vortex finder tube is not coincident with the outer diameter of the separator. This impacts the formation of the inner swirl that flows directly to the vortex finder tube.

The study of the influence of the key geometrical features on the flow structure will be a part of a further investigation, once the numerical model is validated.
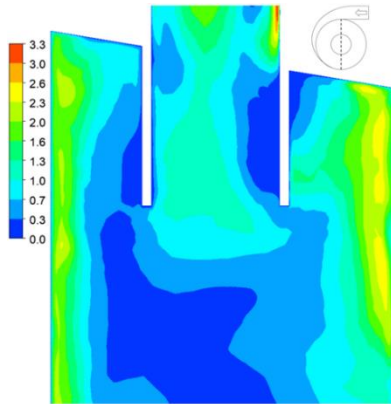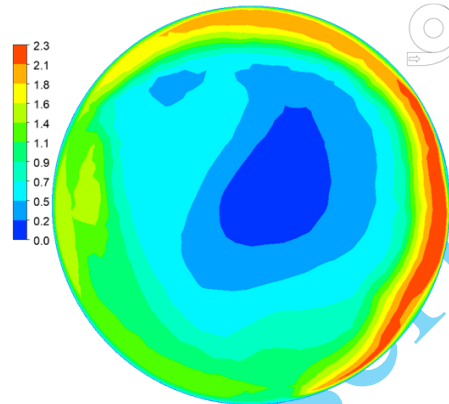

Figure 8. Velocity contour

Figure 9. Velocity contour
(bottom outlet of the separator)

The velocity distributions in the separator are visualized in Figure 8 and Figure 9. The values presented were normalized by the values of the average velocity at the inlet to the cyclone. The areas of both very low speed in the center of the separator (0-0.3) and high speed close to the walls (3-3.3) are observed. The changes in flow parameters indicated that the analyzed flow field inside the separator was unsteady. This is confirmed in Figure 5 where the amplitude of the mass flow rate of oil is depicted.

Table 3. Comparison of simulation results with test bench results

Parameter	Simulation results	Difference to test results
Normalized oil quality [%]	93.8	6.2
Normalized separation efficiency [%]	99.95	0.05

The values of oil quality OQ and separation efficiency η_s which were estimated in the experiment were compared with the results of numerical simulations. The discrepancy in normalized oil quality equals 6.2% and in separation efficiency 0.05% (Table 3). This accuracy can be considered as satisfying.

4. Conclusions

Based on the presented calculation scheme, the separator analysis was performed. The VoF model used for the analysis presented an acceptable level of accuracy when a consensus between accuracy, robustness and time consumption is needed. The selected boundary conditions enabled keeping stable oil volume inside the separator. The pressure drops calculated were comparable to the ones recorded in experimental conditions. The RNG k-ε model shows the creation of a swirl structure inside the core of the separator. As assumed, the calculation shows that oil level in the tank has an impact on the formation of the flow field inside the separator. The separator inlet impacts the formation of the oil fraction close to the wall and its interaction with the free surface of the oil inside the tank. The preliminary results show that the model can give results which are in good alignment with test results. In the next step, the analysis of calculation results for other experimental conditions will be performed.

References

[1]   ARPANDI I., JOSHI A.R., SHOHAM O., SHIRAZI S., KOUBA G.E. 1996. *Hydrodynamics of two-phase flow in gas-liquid cylindrical cyclone separators*. *Society of Petroleum Engineers*. 1, pp. 427–436. doi: 10.2118/30683-PA

[2]   CHANG P., HU T., WANG L., CHANG S, WANG T., WANG Y. 2016. *Numerical simulation on structure optimization of liquid-gas cylindrical cyclone separator. International Journal of Chemical Engineering*. 3187631. doi: 10.1155/2016/3187631

[3]   DELGADILLO J.A., RAJAMANI R.K 2005. *A comparative study of three turbulence-closure models for the hydrocyclone problem. International Journal of Mineral Processing*. 77, 1, pp.217–230. doi: 10.1016/j.minpro.2005.06.007

[4]   EASTWICK C.N, SIMMONS K., WANG Y., 2006. *Study of aero-engine oil-air separators. Journal of Power and Energy*. 220, 7, pp. 707-717.
        doi.org/10.1243/09576509JPE116

[5]   ERDAL F.M., SHIRAZI S.A., MANTILLA I., SHOHAM O. 1998. *CFD Study of bubble carry-under in gas-liquid cylindrical cyclone separators. Society of Petroleum Engineers*. SPE-49309-MS. doi: 10.2118/49309-MS

[6]   ESCUE A., CUI J. 2010. *Comparison of turbulence models in simulating swirling pipe flows. Applied Mathematical Modelling*. 34, 10, pp. 2840–2849.

doi: 10.1016/j.apm.2009.12.018

[7]   FILIPPONE A., BOJDO N. 2010. *Turboshaft engine air particle separation. Progress in Aerospace Sciences*. 46, 5-6, pp.224-245. doi.org/10.1016/j.paerosci.2010.02.001

[8]   GAO X., ZHAO Y., YANG X., CHANG Y., PENG X. 2012. *The research on the performance of oil-gas cyclone separators in oil injected compressor systems with considering the collision and breakup of oil droplets. International Compressor Engineering Conference.* 2119.

[9]   HIRT C.W., NICHOLS B.D. 1981. *Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries*. *Journal of Computational Physics*. 39, pp. 201–255.
      doi: 10.1016/0021-9991(81)90145-5

[10] KRISTOFFERSEN T., HOLDEN C., SKOGESTAD S., EGELAND O. 2017. *Control-Oriented Modelling of Gas-Liquid Cylindrical Cyclones*. *American Control Conference*. pp. 2829–2836. doi: 10.23919/ACC.2017.7963380

[11] UBBINK O. 1997. *Numerical prediction of two fluid systems with sharp interfaces (Phd). Imperial College London*. hdl.handle.net/10044/1/8604

[12] RUDOLF P. 2013. *Simulation of multiphase flow in hydrocyclone*. *EPJ Web of Conferences*. 45, 01101. doi: 10.1051/epjconf/20134501101

[13] SZWARC T., WRÓBLEWSKI W., BORZĘCKI T. 2019. *Numerical simulation of cyclone used in aircraft turbine engine. Modelling in Engineering*. 40, 74.

[14] WANG J., Mao Y., Liu M. 2010. *Numerical simulation of strongly swirling flow in cyclone separator by using an advanced RNG k-ε model. Shiyou Xuebao, Shiyou Jiagong/Acta Petrolei Sinica (Petroleum Processing Section)*. 26, pp. 8-13.

[15] WANG L., GAO X., Feng J.M., Peng X.Y., 2015. *Research on the two-phase flow and separation mechanism in the oil–gas cyclone separator. IOP Conf. Series*: *Materials Science and Engineering*. 90, 012075. doi: 10.1088/1757-899X/90/1/012075

[16] Yakhot V., Orszag S.A., 1986. *Renormalization Group Analysis of Turbulence. I. Basic Theory. Journal of Scientific Computing*. 1, 1. doi: 10.1007/BF01061452