U.P.B. Sci. Bull., Series D, Vol. 83, Iss. 3, 2021 ISSN 1454-2358

NUMERICAL CFD ANALYSIS IN A RADIAL DECANTER

Bianca-Ștefania ZĂBAVĂ[1], Gabriel-Alexandru CONSTANTIN[2], Gheorghe VOICU[3]

Sedimentation is a rather complex process, governed by a series of hydrodynamic phenomena such as: turbulent areas, bottom currents, surface currents, or even temperature fluctuations in the external environment or in the liquid mass. This paper presents the CFD numerical analysis in a radial decanter. Were obtained the maximum values for total velocity (0.93 m/s), average total velocity (0.81 m/s), dynamic pressure (447 Pa), fluid deformation rate (1.39 1/s), turbulent kinetic energy (1.38 m^2/s^2), and turbulence intensity (9.6 %). For the decanter analysed in this paper, which has values of turbulence intensity of up to 9.6%, especially in the water supply area, it is recommended to increase the diameter of the supply pipe in the supply area, but keeping the flow rate.

Keywords: CFD analysis, radial clarifier, turbulence intensity

1. Introduction

Recently, CFD (Computational Fluid Dynamics) analyses are increasingly used in the study of the separation of any gas-liquid-solid mixture, the aim being, in general, to optimize certain equipment that works with such mixtures. It can be said that CFD is actually the science of predicting fluid flow, but also mass and heat transfer. Internationally there are a lot of software that perform such analysis, but the most used (at least in Europe), is Ansys, through the Ansys Fluent, Ansys Polyflow and Ansys CFX tools, [1,2]. Ansys Fluent software uses the finite volume method to solve the integral equations governing the conservation of mass and momentum, but also to determine scales, such as turbulence and solids concentration, [3,4].

In 1977, Larsen developed the first 2D CFD clarifier model in order to simulate the flow field in a rectangular clarifier, a study in which he used a vorticity-flow function formulation in combination with a Prandtl mixing length theory for turbulent modelling, [5-7]. Over the last two decades, advances in CFD have enabled clarifier design optimization based on numerical simulation, [8-11].

[1] Assist., Biotechnical Systems Department, University POLITEHNICA of Bucharest, Romania, e-mail: bianca.dragoiu@yahoo.com
[2] Lect., Biotechnical Systems Department, University POLITEHNICA of Bucharest, Romania, e-mail: gabriel.constantin@upb.ro
[3] Prof., Biotechnical Systems Department, University POLITEHNICA of Bucharest, Romania, e-mail: ghvoicu_2005@yahoo.com

The aim of this study was to simulate the flow of a liquid-solid mixture inside a radial clarifier, in the feeding area. It should be mentioned that a 2D study of the flow analysis was done.

In figure 1 a methodology for analysing the flow of the liquid-solid mixture is presented.

COMPUTATIONAL DOMAIN MODELLING

- 2D modelling in the “Design Modeler” module of the radial decanter by sectioning in its central area; Only the projection of the net volume of the decanter is modelled.

ANALYSIS OF THE LIQUID-SOLID MIXTURE FLOW IN THE RADIAL DECANTER

- Introduction of the drawing (after the mesh of finite volumes has been discretized) in the "Setup" module
- Specifying the mathematical model that governs the flow and specifying the discrete phase (solid) in the main phase (liquid).
- Specifying the materials in the mixture (using the Ansys library)
- Specifying the boundary conditions.
- Specification of calculation methods
- Specifying the time conditions for the calculation (number of calculation steps, size of a step, number of iterative equations solved in a calculation step)
- Rolling the calculus

Fig. 1. Methodology for analysing the flow of the mixture through the radial decanter

The mathematical equations governing this study are presented below, [12-15].

The continuity equation has the form:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{u}) = 0 \tag{1}$$

Momentum equation can be calculated with:

$$\frac{\partial (\rho \vec{\vartheta})}{\partial t} + \nabla \cdot (\rho \vec{u} \vec{u}) = -\nabla P + \rho g + \vec{f} \tag{2}$$

where t is time, ρ is the fluid density, u is the fluid velocity, P is the pressure in the system, f indicates the volume force exerted on the fluid, and ϑ is the dynamic viscosity of the fluid.

The two equations governing the study must be solved simultaneously for the velocity of the fluids in the mixture to be obtained. In a standard mathematical model of analysis, turbulent kinetic energy has the form, [15]:

$$\frac{\partial \rho k}{\partial t}+\frac{\partial(\rho V_x k)}{\partial x}+\frac{\partial(\rho V_y k)}{\partial y}+\frac{\partial(\rho V_z k)}{\partial z} = \frac{\partial}{\partial x}\left(\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x}\right)+\frac{\partial}{\partial y}\left(\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial y}\right)+\frac{\partial}{\partial z}\left(\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial z}\right)+\mu_t\Phi-\rho\varepsilon + \frac{C_4\beta\mu_t}{\sigma_t}\left(g_x\frac{\partial T}{\partial x}+g_y\frac{\partial T}{\partial y}+g_z\frac{\partial T}{\partial z}\right) \quad (3)$$

and the dissipation rate is shaped by, [15]:

$$\frac{\partial \rho \varepsilon}{\partial t}+\frac{\partial(\rho V_x \varepsilon)}{\partial x}+\frac{\partial(\rho V_y \varepsilon)}{\partial y}+\frac{\partial(\rho V_z \varepsilon)}{\partial z} = \frac{\partial}{\partial x}\left(\frac{\mu_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x}\right)+\frac{\partial}{\partial y}\left(\frac{\mu_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial y}\right)+\frac{\partial}{\partial z}\left(\frac{\mu_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial z}\right)+C_{1\varepsilon}\mu_t\frac{\varepsilon}{k}\Phi - C_2\rho\frac{\varepsilon^2}{k}+\frac{C_\mu(1-C_3)\beta\rho k}{\sigma_t}\left(g_x\frac{\partial T}{\partial x}+g_y\frac{\partial T}{\partial y}+g_z\frac{\partial T}{\partial z}\right) \quad (4)$$

But, in this study, a mathematical model based on equations k-ε (turbulent kinetic energy - eddy viscosity) was used, as a result the turbulent kinetic energy equation has the form, [7]:

$$\frac{\partial \rho \varepsilon}{\partial t}+\nabla(\rho v k)=\nabla\left(\frac{\mu_t}{\sigma_t}\nabla k\right)+P_k+G_k-\rho\varepsilon \quad (5)$$

and dissipation rate has the form, [7]:

$$\frac{\partial \rho \varepsilon}{\partial t}+\nabla(\rho v k)=\nabla\left(\frac{\mu_t}{\sigma_t}\nabla \varepsilon\right)+C_{1\varepsilon}\rho\frac{\varepsilon}{k}(P_k+G_k-C_{3\varepsilon}G_k)-C_{2\epsilon}\rho\frac{\varepsilon^2}{k} \quad (6)$$

According to the dictionary of the American Meteorological Society eddy viscosity is defined as the turbulent transfer of impulse by the vortices that give rise to it due to the internal friction of the fluids, in a manner analogous to the action of molecular viscosity in a laminar flow, but everything taking place on a much larger scale, [16].

According to [15], the constants used in the above relations have the values: $C_{1\varepsilon}$=1.42, $C_{2\varepsilon}$=1.68, σ_t=1, C_μ=0.0845.

The trajectory that solid particles dispersed in the primary fluid (water) have, it comes from the balance of forces acting on each particle.

$$\frac{du_p}{dt}=F_D(u-u_p)+\frac{g_x(\rho_p-\rho)}{\rho_p}+F_x \quad (7)$$

where F_x is an additional term for acceleration, and $F_D(u-u_p)$ represents the drag force exerted on the particle.

The drag force of the particle is determined by the relationship:

$$F_D=\frac{18\mu}{\rho_p d_p^2}\frac{C_D Re}{24} \quad (8)$$

where u, u_p, μ, ρ, ρ_p and d_p represents the velocity of the fluid phase, particle velocity, fluid viscosity, fluid density, particle density and diameter.

The relative number of Reynolds can also be calculated with the relation:

$$Re = \frac{\rho d_p (u - u_p)}{\mu} \tag{9}$$

According to the literature [12], the coefficient of drag of the particle in the primary fluid, C_D (from ec. (8)), falls within the Eulerian-Lagrangian reference. This coefficient changes with the variation of the Reynolds number in each finite volume in which the calculation is made, while the geometry of the computational domain, as well as the particle diameter remain constant throughout the process.

$$C_{D,sferă} = \begin{cases} 0{,}424, Re > 1000 \\ \dfrac{24}{Re}\left(1 + \dfrac{1}{6} Re^{2/3}\right), Re < 1000 \end{cases} \tag{10}$$

The additional term for the acceleration of particles in the fluid, appearing in equation (7) and which is actually a fluid gradient of virtual weight force and pressure, can be calculated with the relationship:

$$F_x = C_{vm} \frac{\rho}{\rho_p} \left(\overrightarrow{u_p} \nabla \vec{u} - \frac{d\overrightarrow{u_p}}{dt} \right) \tag{11}$$

where C_{vm} represents the virtual weight coefficient, usually set to 0.5 in such analyses, [14].

The net force resulting from equation (7) must be equal to the inertia of the particle along the OX axis of the Cartesian system in which the geometric pattern is drawn.

2. Material and method

For this purpose, a laptop was used on which the Ansys 19.2 software was installed. A working session with the Ansys Fluent tool was opened in the program workbench.

The operations necessary to perform the analysis are: realization of geometry, realization of the finite volumes mesh, specification of the characteristic elements of the analysis, exploration of possible solutions and finally visualization of the analysis results. All these steps were taken in order to perform the numerical study.

Before starting the study, several simplifying hypotheses were decided:

- the viscosity and density of the fluids are constant throughout the process;
- fluid velocity at the wall is zero;
- the velocity is constant along the entire length of the supply area;
- the water discharge area from the decanter was defined in the boundary conditions of the computational field as a discharge area (outflow);
- solid particles dispersed in the primary fluid (water) are defined with the discrete phase option and all have a spherical shape;

- the analysis will be done 2D in the middle area of the radial decanter.

After establishing the steps to be followed, the geometric modelling of the 2D view of the decanter was started.

Figure 2 shows a simplified 3D modelling of the radial decanter, and in fig. 3 shows a section through the central area of the decanter, section passing through the middle area of the supply pipe. Both views were made in the SolidWorks 2016 SP 0.0 parameterized design software.

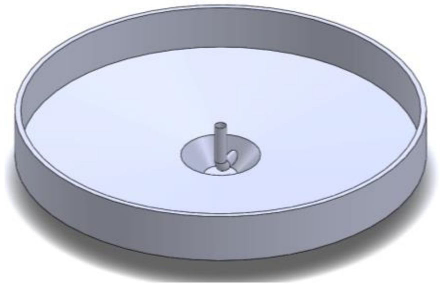

Fig. 2. Isometric view of a radial decanter in simplified structure

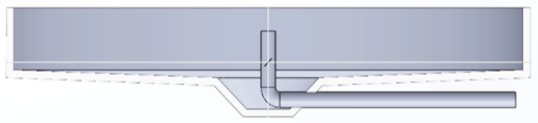

Fig. 3. Section in the central area of the decanter

Having an overview of the projection of the net volume of the decanter, in the central area, the 2D geometric model used for the analysis in Ansys 19.2 was made. The geometric model was made in the "Design Modeler" module and is presented in fig. 4.

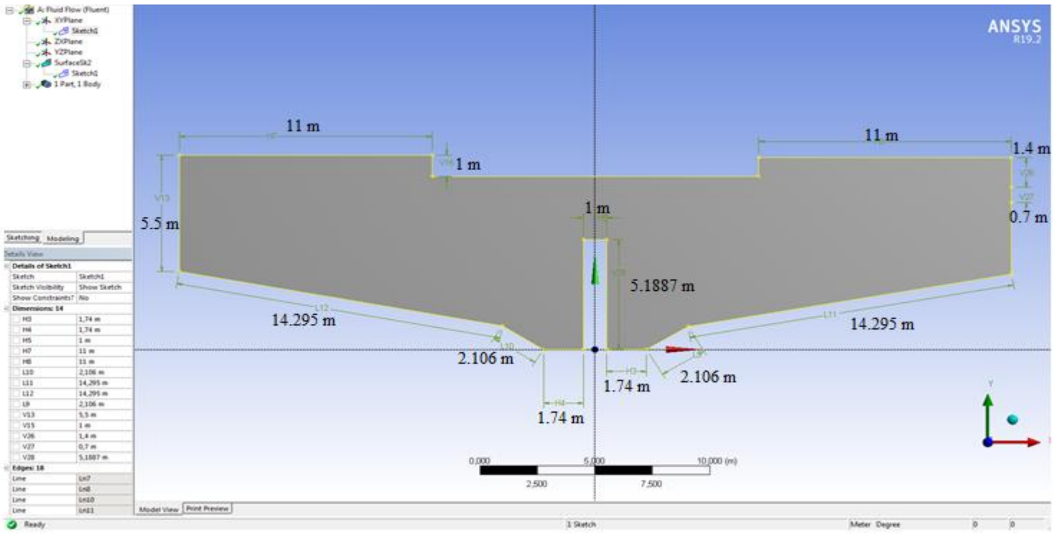

Fig. 4. Geometric model of the decanter made for analysis

The outline of the sketch was made on the XOY plane, the sketch was dimensioned, and finally it was defined as a virtual volume by means of the surface tool in the sketch („surfaces from sketches”). It should be noted that the sketch is in fact a 2D view of the projection of the net volume of the decanter in the central area.

One of the most important steps in performing an analysis is to generate the finite volume mesh. The accuracy of the analysis results depends very much on its finesse, but also the convergence of the results in certain areas of the geometry under analysis.

The finite volumes mesh was made in the "Mesh" module of the program and a different type of mesh realization was chosen than the standard one. Thus, the geometric model was divided into 6 groups (named by the program Edge Sizing), each containing a certain number of edges, groups to which a certain number of finite volume divisions have been set. So, for Edge Sizing 1 200 divisions were set, for Edge Sizing 2 – 150, Edge Sizing 3 – 70, Edge Sizing 4 – 250, Edge Sizing – 35 and Edge Sizing 6 – 100, all these groups actually delimiting the contour of the geometric model. By imposing a large number of finite volume divisions, a fairly high mathematical model accuracy was actually ensured in these areas, considered delicate. Then, the surface of the geometric model was divided into finite volumes by the triangulation method and using as order for elements the Quadratic option. After mesh realization (which took about 60 minutes), a mesh with 565714 elements and 1142589 nodes was obtained, see fig. 5.

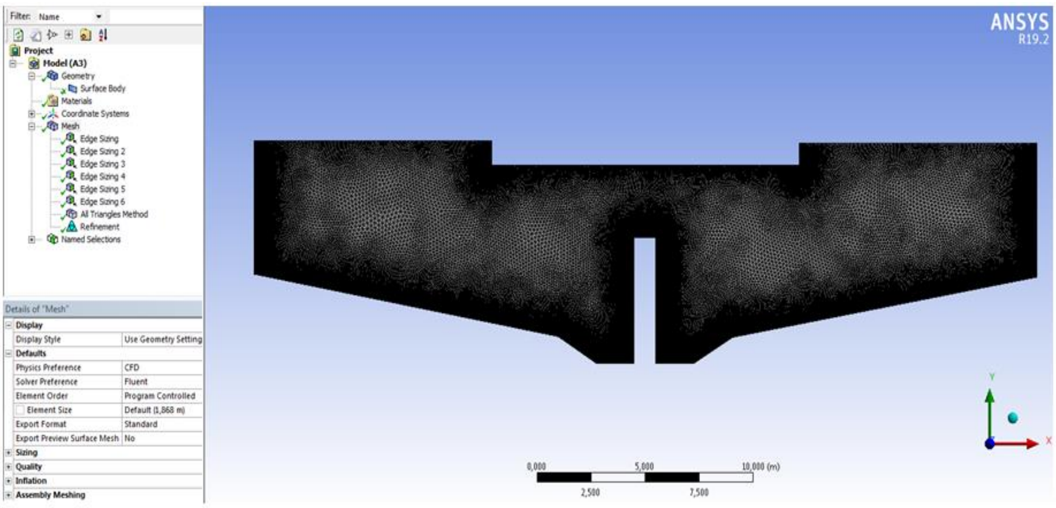

Fig. 5. The mesh of finite volumes obtained for the geometric model

After discretization, the water supply area and the water outlet area were set as separate areas (fig. 6).

Having the CFD option as a physical reference, all cells in the discretized network were automatically set to Fluid.

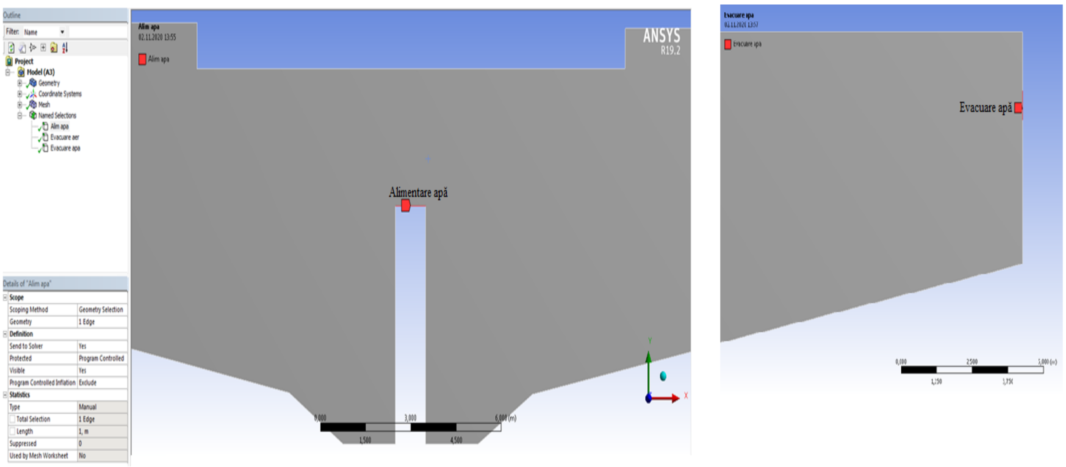

Fig. 6. Water supply and water outlet area

Further, the discretized geometric model was loaded into the "Setup" module to establish the analysis conditions. Time was first set as Transient to allow particles to move through the fluid mass, and the gravitational acceleration was set on the OY axis with value -9.81 m/s^2 because the water supply is in the opposite direction to the gravitational force.

Knowing the diameter of the supply pipe in the decanter (1 m), water density (1000 kg/m^3), water viscosity ($8.9 \cdot 10^{-4}$ m²/s) and the maximum water speed in the velocity area (0.036 m/s according to [17]), it was possible to calculate the Reynolds number in the supply area in the decanter. This is equal to 40449.44, which frames the flow regime as turbulent. As a result, the mathematical model k-ε (2 equations) was chosen to substantiate the study. To increase the accuracy of the study was selected as a model k-ε the option RNG (Re-Normalisation Group). This is a method of renormalizing the Navier-Stokes equations to explain the effects of smaller scales of motion and was first proposed in the paper [18]. In the standard model k-ε, the eddy viscosity is determined from a single turbulence length scale, so that the calculated turbulent diffusion is that which occurs only at the specified scale, while in reality all scales of motion will contribute to turbulent diffusion. RNG approach, which is a mathematical technique that can be used to obtain a turbulence model similar to k-ε, results in a modified form of the epsilon equation that tries to explain the different scales of motion by changes in the production term, [19].

The behavior of the fluid near the walls was set as standard, and the constants in the equations k-ε have been set as shown above.

With the choice of the mathematical model, particle injection was also required. As a result, the discrete phase has been activated (Discrete Phase – On), the material to be injected into the water supply area has been chosen (calcium carbonate), the injection position has been set, particle diameter (max. 0.001 m), flow rate (0.1 kg/s), the time interval for the injection (from 0 to 10 s simulation time) and collision behaviour. The primary fluid was then set by choosing the water-liquid option from the Ansys software library.

Next, was proceeded to establish the boundary conditions of the simulation. Thus, the water supply area has been set as the supply region with the velocity-inlet option and it has established a primary fluid supply velocity of 0.036 m/s, and the water outlet area has been set as the outlet area with the outflow option. The operating conditions (pressure and gravitational acceleration) were also set here.

Subsequently, the method of analysis was chosen. Thus, for the pressure, momentum, turbulent kinetic energy and dissipation rate, the second order equations were selected in order to obtain the highest possible calculation accuracy.

After initializing the calculus, the calculation parameters were set and the calculation was run. The step size of the calculation has been set to 0.005 s, the number of steps to 2000 and the number of iterations per time step at 20. By multiplying the size of the calculation step (0.005 s) with the number of steps (2000) the total simulation time is obtained (10 s). The calculation was started by pressing the Calculate button.

During the calculation, the software draws in real time the curves of variation of the residual values. It should be noted that the total calculation time was 26.5 hours.

A residual value is defined as the difference between the actual value and the forecasted value, [20]. The closer the residual values are to zero, the more convergent the results. In this analysis it can be seen that the residual values have reached a value of 10^{-5} which is quite satisfying.

3. Results and discussions

After completing the calculation, the simulation results were visualized. Ansys 19.2 software also presents the results in the form of coloured contours, the value of each coloured region being given in a chromatic legend, but also in the form of charts or histograms. The results obtained are presented below. It should be noted that the maximum values appear in the water supply area, as well as in the outlet area.

In figure 7 the distribution of the dynamic pressure in the liquid mass is presented. It can be easily seen that the maximum values are reached in the feeding area inside the decanter, absolutely logical thing anyway, because in this area the highest deformation rate is also reached (fig. 8).

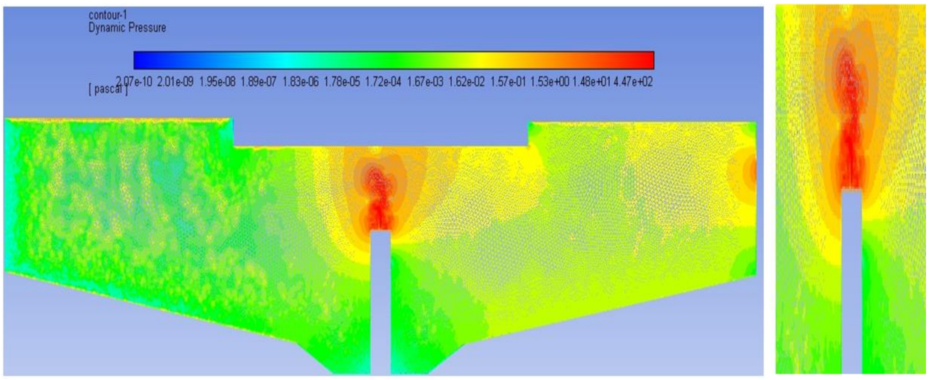

Fig. 7. Dynamic pressure

The pressure distribution is oriented towards the outlet area of the decanter, the liquid moving more in this direction.

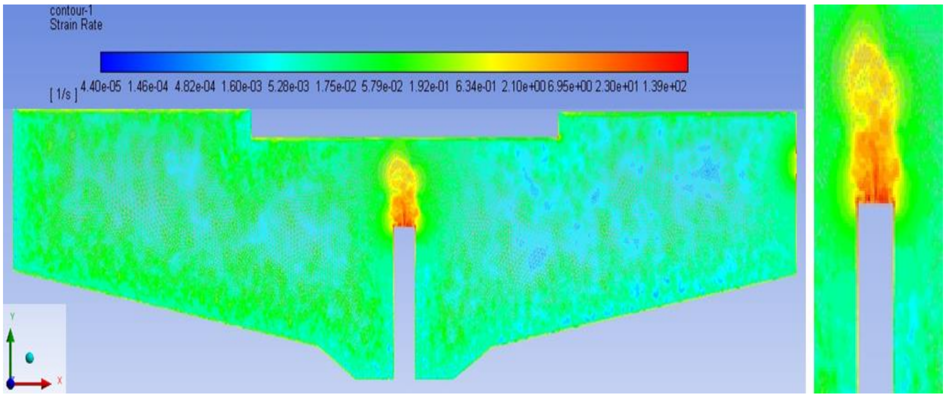

Fig. 8. Fluid strain rate at 10 seconds of simulation

Maximum fluid velocity values are obtained in the same areas (water supply area and outlet area), generally due to deformation of the liquid. Fig. 9 shows the velocity magnitude distribution in the supply area

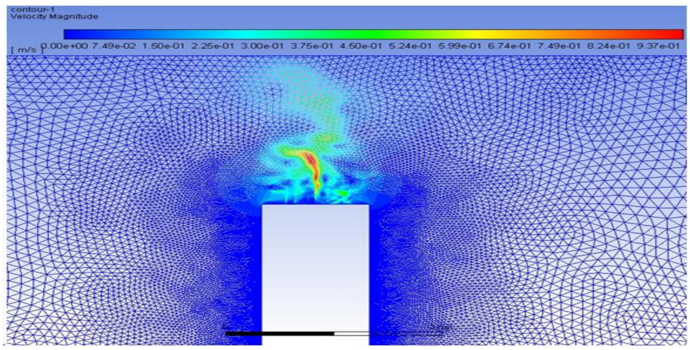

Fig. 9. Velocity magnitude

Figure 10 shows the mean velocity magnitude along the length of the radial decanter.

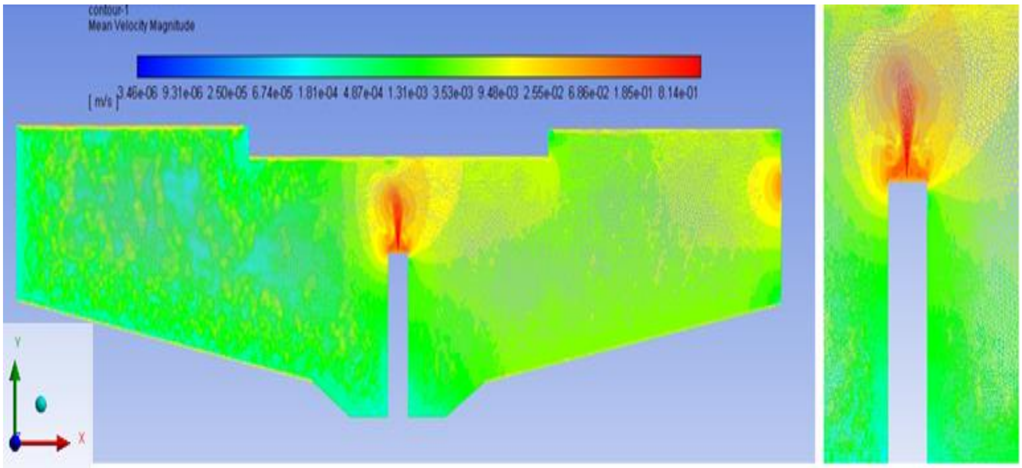

Fig. 10. Mean velocity magnitude

The distribution of turbulent kinetic energy is shown in fig. 11. With its help, the areas with the highest potential for hydrodynamic losses can be observed, losses that favour the sedimentation process. In order to produce as few vortices as possible it is desirable that the sedimentation process be carried out in the presence of a turbulent kinetic energy, of the lowest possible value.

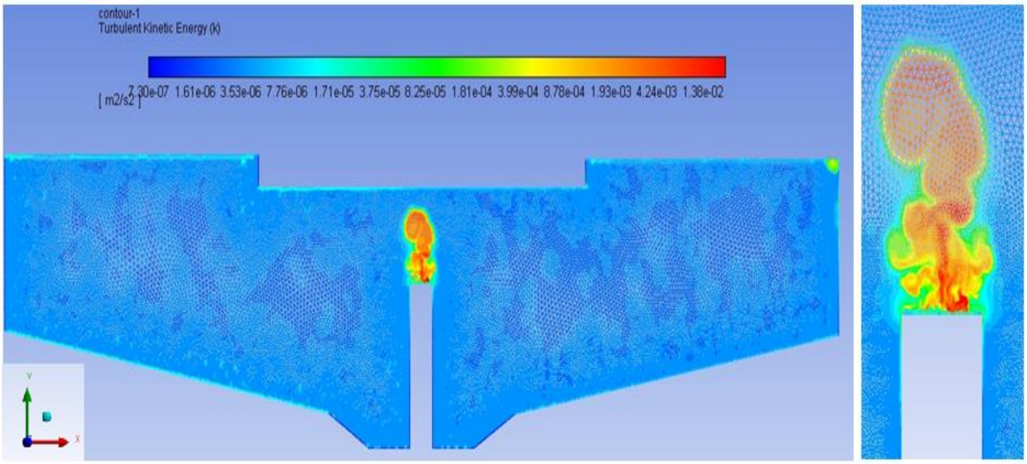

Fig. 11. Turbulent kinetic energy

The distribution of turbulence intensity is shown in fig. 12. The maximum percentage values (9.6%) are also obtained in the supply area.

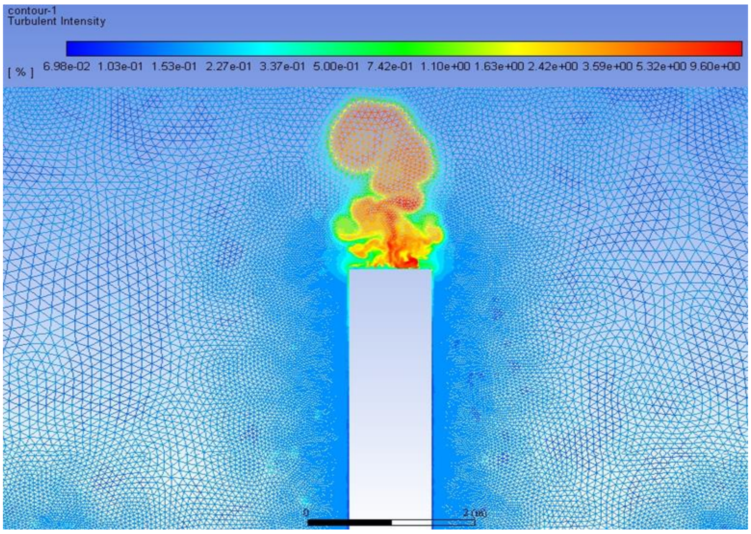

Fig. 12. Turbulence intensity

4. Conclusions

In the paper, the numerical CFD analysis in a radial decanter was performed.

This type of study should be performed for any constructive variant of decanters. It is quite important to identify areas of turbulence. In these areas the velocity of the fluid, dynamic pressure in fluid, but also its strain rate increases, which leads to a propagation of turbulence throughout the mass of liquid. As the rate of occurrence of vortices in the mass of liquid is greater, thus decreasing the probability of settling those heavy particles that should settle as quickly as possible at the bottom of the decanter.

For the decanter analysed in this paper, which has values of turbulence intensity of up to 9.6%, especially in the water supply area, it is recommended to increase the diameter of the supply pipe in the supply area, but keeping the flow rate. By increasing the diameter only in the supply area, lower liquid flow velocity will be obtained, which will reduce the likelihood of vortices occurrence.

R E F E R E N C E S

[1].***ANSYS, Computational Fluid Dynamics (CFD) Simulation https://www.ansys.com/products/fluids.

[2]. *** ANALYSIS TOOLS, ANSYS Advantage - Volume II, Issue 4, 2008.

[3]. *A. H. Ghawi and J. Kriš*, "A Computational Fluid Dynamics Model of Flow and Settling in Sedimentation Tanks", Applied Computational Fluid Dynamics, Prof. Hyoung Woo Oh (Ed.) , InTech, March 2012, pp. 19-34, ISBN: 978-953-51- 0271-7.

[4]. *H. Gao and M. K. Stenstrom*, "Evaluating the Effects of Inlet Geometry on the Limiting Flux of Secondary Settling Tanks with CFD Model and 1D Flux Theory Model", in J. Environ. Eng., **vol. 145,** nr. 10, 2019.
[5]. *P. Larsen*, "On the hydraulics of rectangular settling basins, experimental and theoretical studies". Report no. 1001, in Dept of Water Resour Engrg, Lund Inst of Tech, Lund Univ, Lund, Sweden, 1977.
[6]. *S. Das, H. Bai, C. Wu, et al.*, "Improving the performance of industrial clarifiers using three dimensional computational fluid dynamics",in Eng. Appl. Comput. Fluid Mech., **vol. 10**, nr. 1, 2016, pp. 130-144, DOI: 10.1080/19942060.2015.1121518.
[7]. *M.E. Valle Medina*, "Secondary settling tanks modeling : study of the dynamics of activated sludge sedimentation by computational fluids dynamics", Chemical and Process Engineering. Université de Strasbourg, 2019.
[8]. *A.A. Al-Jeebory, J. Kris and J.H. Ghawi*, "Performance improvement of water treatment plants in Iraq by CFD model", in Al-Qadisiya Journal For Engineering Sciences, **vol. 3**, nr. 1, 2010, pp. 1–13.
[9].*A.G. Ghawi and J. Kriš*, "Improvement performance of secondary clarifiers by a computational fluid dynamics model", in SJCE, **vol. 19,** nr. 4, , 2011, pp. 1–11.
[10]. *K. Ramalingam, S. Xanthos, M. Gong, et al.*, "Critical modeling parameters identified for 3D CFD modeling of rectangular final settling tanks for New York city wastewater treatment plants", in Water Sci. Technol., **vol. 65**, nr. 6, 2012, pp. 1087–1094.
[11]. *D. Lo*, New flocculation centre well design for circular final clarifiers at the Tai Po Sewage Treatment Works", in HKIE Transactions, **vol. 20**, nr. 4, 2013, pp. 221–229.
[12]. *Kh. Sharifi, T. Jafari Behbahani, S. Ebrahimi, et al., "*A new computational fluid dynamics study of a liquid-liquid hydrocyclone in the two phase case for separation of oil droplets and water", in Braz. J. Chem. Eng., **vol. 36**, nr. 4, Oct./Dec. 2019, pp. 1601-1612, https://doi.org/10.1590/0104-6632.20190364s20170619.
[13]. *Andrew Parry , "*Numerical Simulations and Optimisation of Gas-Solid-Liquid Separator", MSc Thesys in Petroleum Engineering, , Imperial College London, 2014.
[14]. *Peter Kohnke, "*ANSYS Theory Reference", Eleventh Edition, SAS IP, Inc, 1999. http://research.me.udel.edu/~lwang/teaching/MEx81/ansys56manual.pdf.
[15]. *Daniel Brennan*, "The numerical simulation of two-phase flows in settling tanks", PhD Thesys, University of London, 2001.
[16]. *** American Meteorological Society (AMS) - Glossary of Meteorology, 2012.
[17]. *H.A.M. Heikal, A.A. El-Hafiz, A.R. El Baz, et al.*, "Study the performance of circular clarifier in existing potable water treatment plant by using computational fluid dynamics", ,in IWRA, XVI World Water Congress Cancun, Quintana Roo. Mexico, 29 May – 3 June, 2017.
[18]. *V. Yakhot, S.A. Orszag, S. Thangam, et al.*, "Development of turbulence models for shear flows by a double expansion technique", in Phys. Fluids., **vol. 4**, no. 7, 1992, pp. 1510-1520.
[19]. *** https://www.cfd-online.com/Wiki/RNG_k-epsilon_model
[20]. *** https://stattrek.com/regression/residual-analysis.aspx