CFD ANALYSIS OF THE SETTLING PROCESS IN A RADIAL CLARIFIER
/
ANALIZA CFD A PROCESULUI DE DECANTARE ÎNTR-UN DECANTOARE RADIAL

Gabriel Alexandru CONSTANTIN [1)], Bianca Stefania ZABAVA [*1)], Gheorghe VOICU [1)], Georgiana MOICEANU [2)], Irina Aura ISTRATE [1)], Mihaela NITU [3)]

[1)] National University of Science and Technology "POLITEHNICA" Bucharest, Faculty of Biotechnical Systems Engineering / Romania;
[2)] National University of Science and Technology "POLITEHNICA" Bucharest, Faculty of Entrepreuneurship, Business Engineering and Management / Romania; [3)] INMA Bucharest / Romania
Tel: 0040731538941; E-mail:bianca.dragoiu@upb.ro
DOI: https://doi.org/10.35633/inmateh-70-15

Keywords: *Computational Fluid Dynamics, fluid velocity, radial clarifier, Reynolds number*

ABSTRACT
The objective of the present study was to make a theoretical study based on a CFD analysis for a conventional radial clarifier. The parameters of the Computational Fluid Dynamics analysis were set in the Ansys software, and after running the simulation, the values for fluid velocity, turbulence intensity and Reynolds number were obtained. Thus, it was obtained a fluid velocity of 0.103 m/s, a turbulence intensity of $3.82 \cdot 10^{-2}$ % and a Reynolds number of 14.7. This work can help researchers in the field, mainly, but also radial clarifier manufacturers to optimise the process.

REZUMAT
Obiectivul prezentului studiu a fost de a realiza un studiu teoretic bazat pe o analiză CFD pentru un clarificator radial convențional. Parametrii analizei dinamicii fluidelor computaționale au fost setați în software-ul Ansys, iar după rularea simulării, au fost obținute valorile pentru viteza fluidului, intensitatea turbulenței și numărul Reynolds. Astfel, s-a obținut o viteză a fluidului de 0,103 m/s, o intensitate a turbulenței de $3,82 \cdot 10^{-2}$ % și un număr Reynolds de 14,7. Această lucrare poate ajuta cercetătorii din domeniu, în principal, dar și constructorii de clarificatoare radiale în sensul optimizării procesului.

INTRODUCTION
Our planet has witnessed an escalating rate of industrialization, urbanization, and population increase over the past 50 years, which has had a negative influence on the quality of the world's water, air, and soil as well as on environmental degradation. Since it promotes sustainable water reuse while decreasing or eliminating pollution of natural water sources, wastewater treatment has become one of society's most important environmental problems (*Alsina, 2008*).

Current wastewater management techniques result in inefficient nutrient recovery and reuse, which can have negative effects on the ecosystem, including eutrophication, the climate, and world food security *(Hoffmann et al., 2020; Öberg et al., 2020).* Water issues are given a lot of attention in a circular economy, which entails more sustainably managing waste and raw materials (including water) (including wastewater) (*Smol, 2022; Smol and Koneczna, 2021).*

In recent decades, water and wastewater treatment plants have attracted the government's attention, especially through the dangers of wastewater pollution from urban areas *(Chero et al., 2019).* Most municipal wastewater treatment facilities are required by law to provide some type of treatment for all flows that enter their facilities, regardless of volume or duration (*Clarifier Design, 2005*).

In Romania, there is the problem of rainwater collection and reuse without being collected in wastewater treatment stations.

Solids removal is probably the most widely used method of water purification from primary mechanical treatment in wastewater treatment plants. A crucial phase in this process refers to the separation of sludge and suspended solid particles using gravity, a process called sedimentation, the equipment in which the process is carried out is called a clarifier. In these tanks, the wastewater is admitted to the tank at one end and the clarified water is discharged at the other end of the clarifier. For the correct deposition of particles, water should pour into the reservoir for enough time (*Hasim et.al., 2020*).

To eliminate suspended solids from influent raw wastewater, primary clarifiers are frequently used in wastewater treatment facilities (*Gernaey and Vanrolleghem, 2005*).

Primary clarifiers, as the first stage of treatment, have an impact on the following biological and sludge treatment units as well as the production of biogas and electric energy in systems with anaerobic digestion and cogeneration.

As a result, primary clarifiers have a significant impact on the efficiency of the wastewater treatment facility as a whole (Patziger *et.al., 2016; Griborio et.al., 2021*).

In the general case, several distinct zones can be identified in a clarifier: the inlet zone, the sedimentation zone, the sediment accumulation zone (sludge zone) and the sediment discharge zone *(Rus, 2001).* The clarifier, as a major installation in wastewater treatment plants, can limit or define the performance of the treatment plant (Das et.al., 2016). In general, up to 50% of the total pollutant load in wastewater is removed by sedimentation *(Manuals of British Practice in Water Pollution Control, 1980).*

Many different things can have an impact on how well a clarifier is performing to illustrate this, the Reynolds number, the viscosity of the water, the type of movement of the water flow, and also the size and construction of the clarifier are the most important factors in the sedimentation unit *(Chero et.al., 2019; Campbell and Empie, 2006)*.

Conventional clarifier models make up about one-third of the total cost of capital water treatment facilities because of the cost of land and construction. Numerous techniques have been developed to increase the effectiveness of tailing ponds, increase their hydraulic capacity, and reduce construction or operational costs (*Saady, 2012).* Recently, Computational Fluid Dynamics (CFD) analyses have become fast and easy to use. These new generation analyses offer a cheap means of testing and optimizing the hydraulic operation of both existing and design constructions *(Al-Jeebory et al., 2010).* CFD is a quick, low-cost method for assessing engineering systems that are difficult to replicate in a lab or real-world situations. This gives it several benefits over traditional modelling techniques. Construction of more efficient and compact sedimentation tanks for conventional water treatment facilities can relate to a famous example of how CFD can build a "virtual prototype". Using CFD, it is possible to visualize the three-dimensional liquid flow inside a tank, which can increase solids separation and decrease turbulence.

The objective of the present study is to make a theoretical study based on a CFD analysis for a conventional radial clarifier. The parameters of the Computational Fluid Dynamics analysis were set in the Ansys software, and after running the simulation, the values for fluid velocity, turbulence intensity and Reynolds number were obtained.

MATERIALS AND METHODS

This study builds on earlier research that was done to optimize the sedimentation process (*Zăbavă et al., 2021)*. The research started with the simulation of the flow of a liquid-solid mixture in a radial clarifier, with a central supply area, made in ANSYS CFX. It is important to note that a 2D flow analysis inquiry was carried out, because the clarifier is radial, and a symmetry of the results appears. The preceding study's mathematical equations and simplifying hypotheses were applied to the modelling *(Zăbavă et al., 2021; Brennan, 2001; Kohnke, 1999; Parry, 2014; Sharifi et al., 2019).*

It should be noted that was run one simulation for a usual clarifier in which the supply pipe had diameter=1 m over, height=5.5 m (the usual case). A simplified 3D modelling of the radial clarifier was performed (Fig. 1), but also a section through the central area of the clarifier, a section that also passes through the middle area of the supply pipe (Fig. 2), using SolidWorks 2016 SP 0.0.

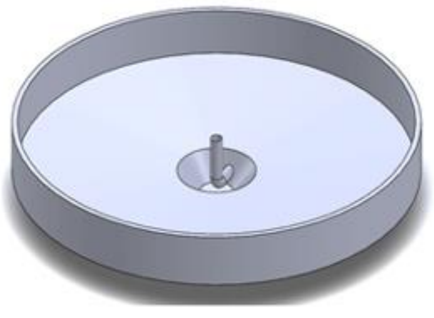


Fig. 1 - Isometric view of a radial clarifier, designed in SolidWorks: 1 m supply line
(Zăbavă et al., 2021)

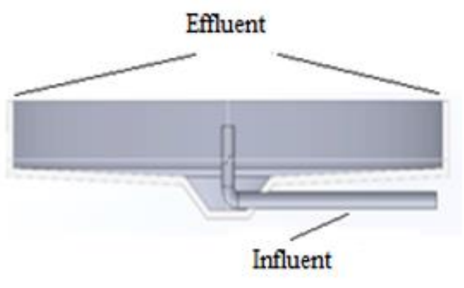


Fig. 2 - Section in the central area for the clarifier with 1 m supply line
(Zăbavă et al., 2021)

The model was drawn in the “Design Modeler” module and are presented in Figure 3.

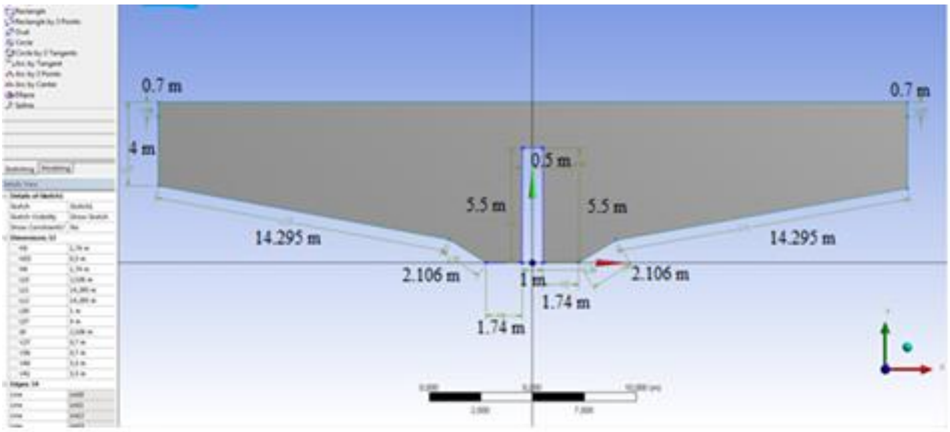


Fig. 3 - Geometric model drawn for the analysis of the clarifier with a feeding pipe with 1 m supply line *(Zăbavă et al., 2021)*

Therefore, 250 divisions have been set for Edge Sizing and for Edge Sizing 2 - 300, basically both groups define the geometry of the model. In these areas, a reasonable level of numerical prediction accuracy was effectively ensured by imposing a considerable number of finite volume divisions, considered delicate for the 1 m feed pipe clarifier. The surface area of the geometric models was then divided into finite volumes using the triangulation method, Quadratic option. Finally, the grid was given a smoothness of grade 3.
After grid generation (which took about 60 minutes), a grid with 372395 elements and 729904 nodes was obtained for the clarifier with 1 m feed pipe (Figure 4).

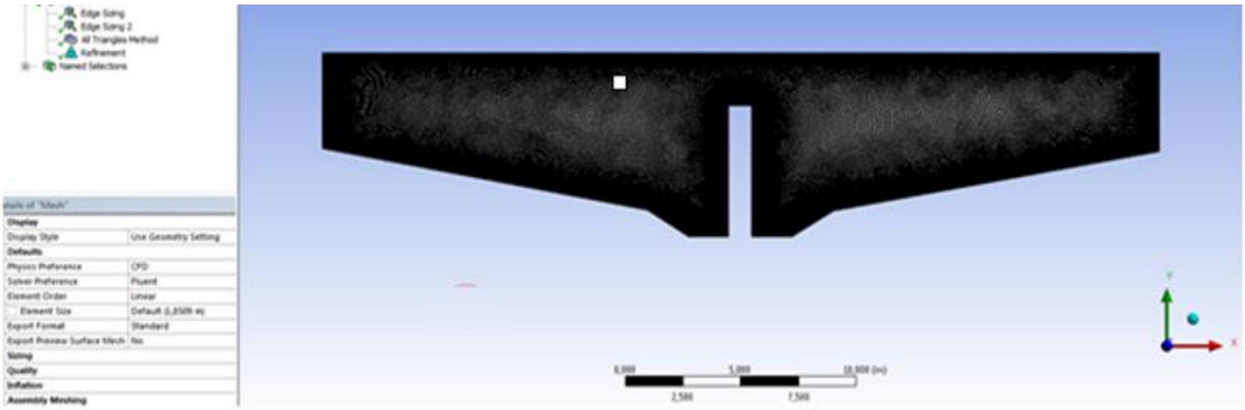


Fig. 4 - Finished volume mesh obtained for the geometric model with 1 m supply line *(Zăbavă et al., 2021)*

Following discretization, the water feed zone and the water outlet zone were configured as independent zones (Figure 5).

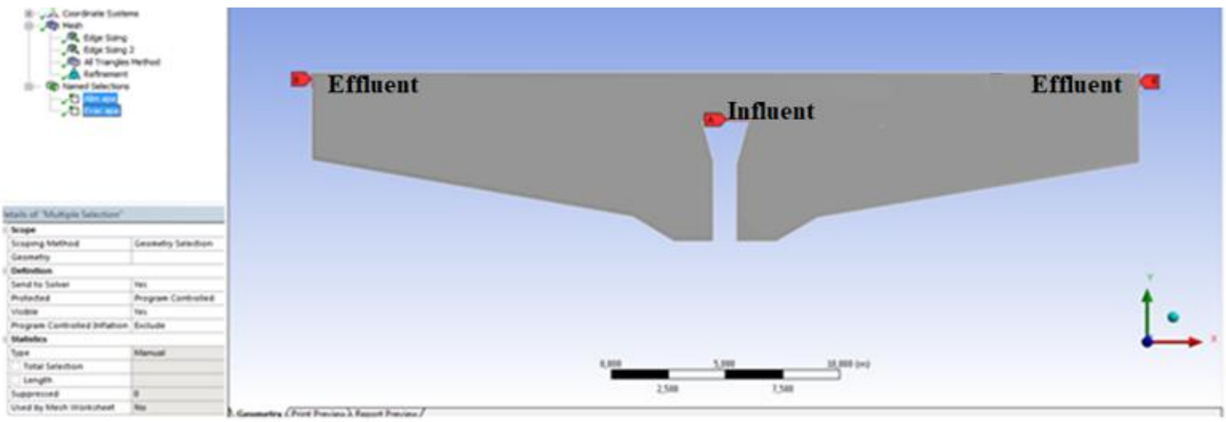


Fig. 5 - Water supply and discharge areas for the clarifier *(Zăbavă et al., 2021)*

With the CFD option as the physical reference, the discretized network cells were automatically assigned to be fluids. Knowing all the values involved in the relation of the Reynolds number, its calculation was performed for the two geometric models, using the calculation formula:

$$Re = \frac{v \cdot \rho \cdot d}{\vartheta} \quad (1)$$

It has a value of 40449.44 for this clarifier, which frames the flow regime as a turbulent one. In order to support the study, the mathematical model k- (2 equations) was used. The RNG (Re-Normalization Group) option was chosen as the k-model to improve study accuracy. This approach to renormalizing the Navier-Stokes equations to account for the effects of motion at lower scales was initially put forth in the publication (*Yakhot, 1992*).

The calcium carbonate to be injected in the water supply area, the injection position, the particle diameter (0.001 m), the flow rate (0.1 kg/s), the time to inject (from 0 to 180 s simulation time), the collision behaviour, and the simulation boundary conditions were all set before the discrete phase was activated. With the velocity-inlet option and a primary fluid feed rate of 0.036 m/s, the water supply area has been configured as the supply region, and the water discharge area has been configured as the outflow. At this point, the operating conditions (pressure and gravitational acceleration) were also established. The analysis approach was then chosen (Figure 6).

The second order equations for pressure, impulse, turbulent kinetic energy, and dissipation rate were selected to achieve the highest calculation accuracy.

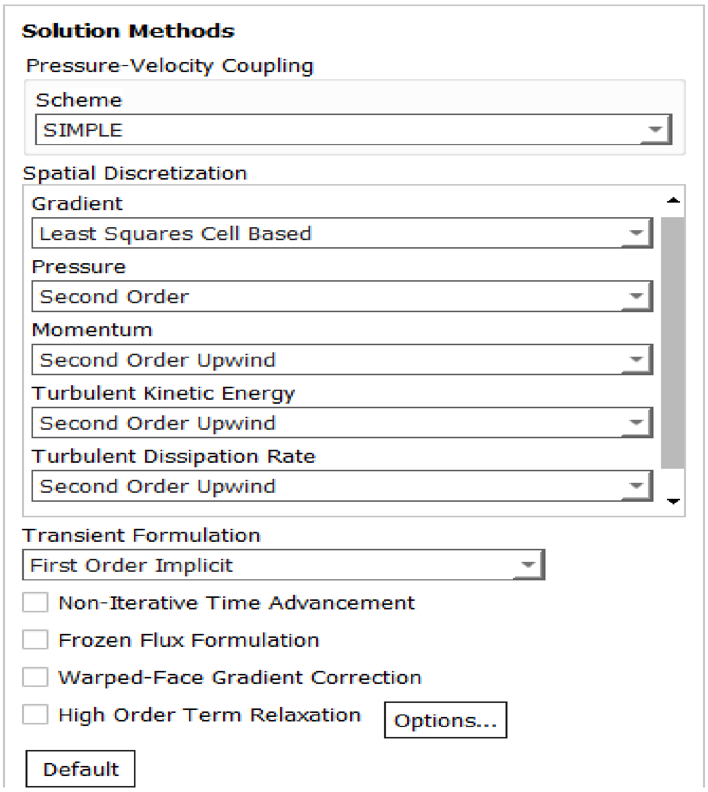


Fig. 6 - Choosing the analysis method in the Ansys program

The following settings were used to finish the calculation: Step size for the calculation was 0.005 seconds, there were 36,000 steps total, and there were 20 iterations for each time step. By dividing the total number of steps (36,000) by the calculation step size (0.005 s), the overall simulation duration (180 s) was determined. The calculation was started by clicking the "Calculate" button. While executing the calculation, the software simultaneously plots the residual value variation curves (Figure 7).

According *to ANALYSIS TOOLS, (2008; Stat Trek)*, this analysis's residual values attained a value of 10^{-4} for this type of clarifier.

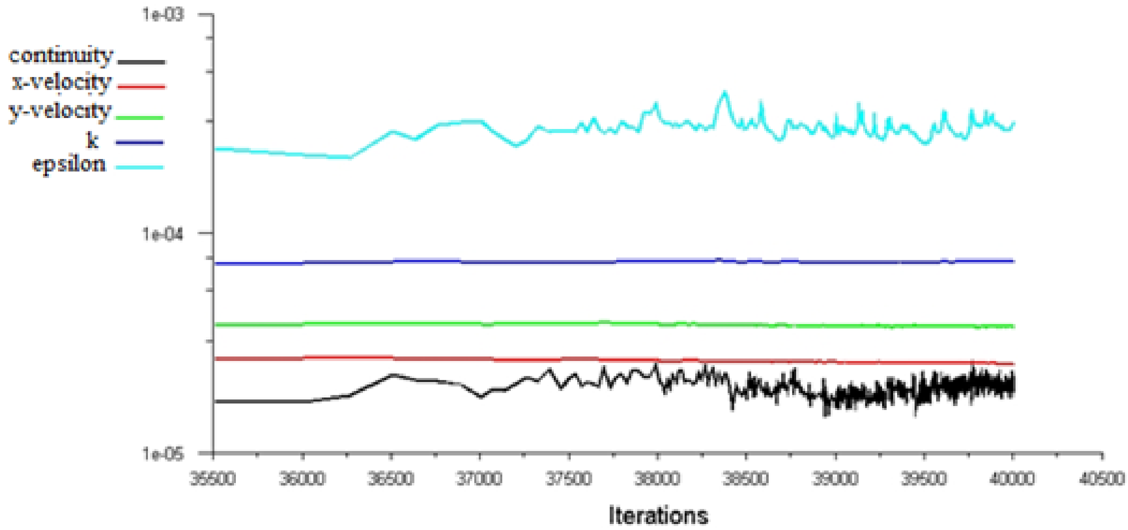


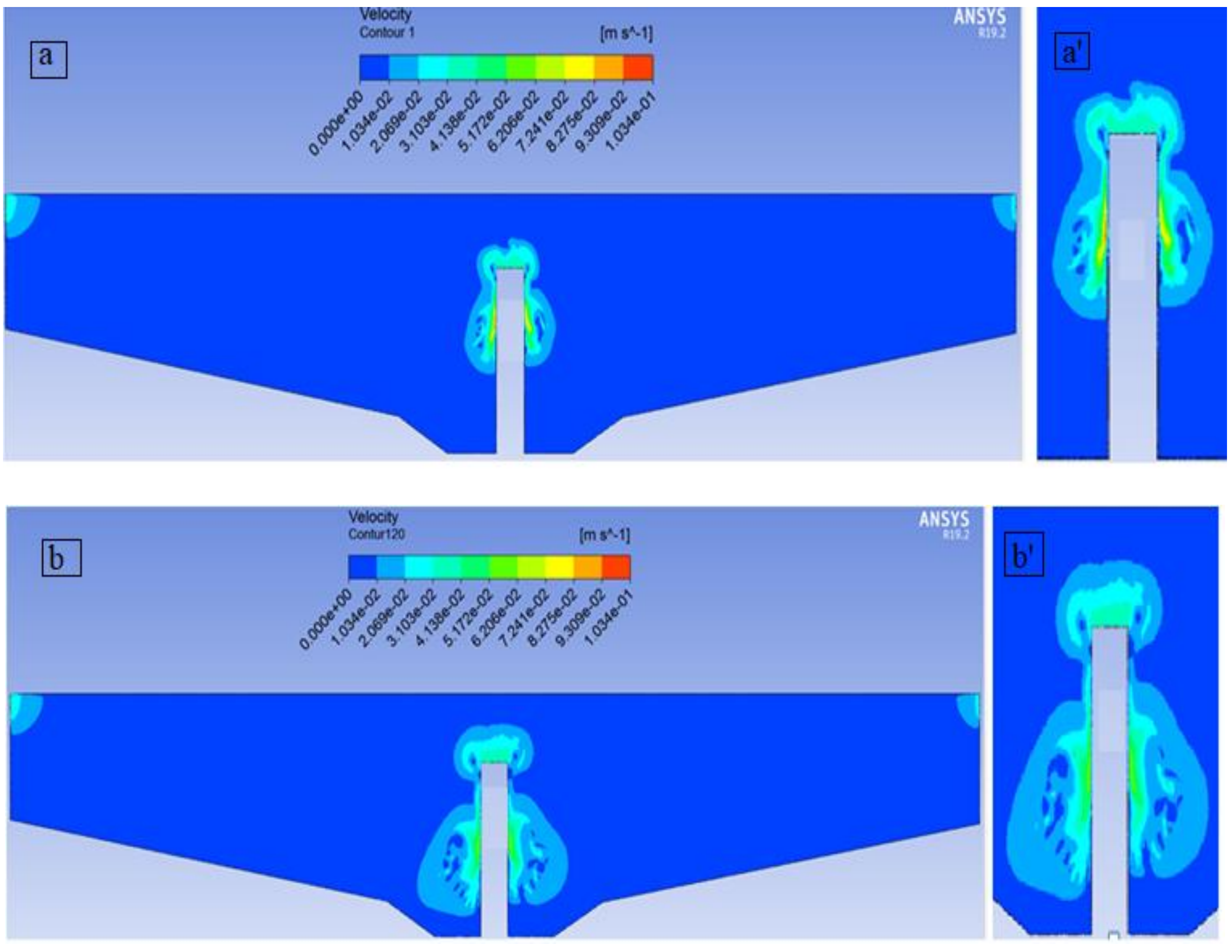
Fig. 7 - Residual values variation curves

RESULTS

The data for velocity, the Reynolds number in each cell, and the intensity of the turbulence at 60 s, 120 s, and 180 s (the simulation time) are reported in this section. It should be noted that a study on a few of the results reported below was conducted by one of the authors of this paper and published in the Ph.D. thesis.

Fluid velocity analysis

Figure 8 displays the velocity distribution for the clarifier with a feeding pipe of 1 m to 60 s, 120 s, and 180 s, as well as specifics of the feeding in those same timeframes.



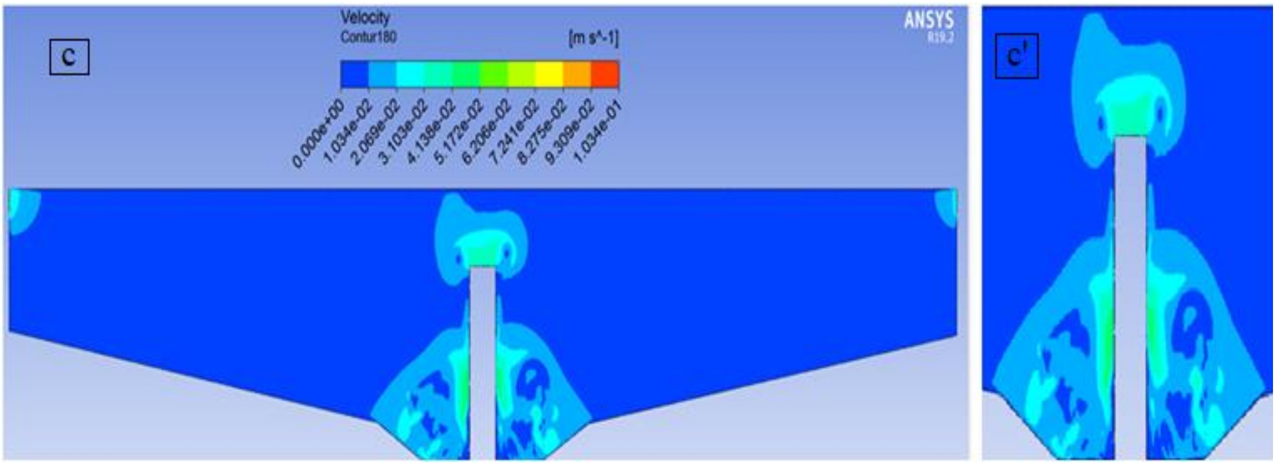


Fig. 8 - Clarifier velocity distribution at 60 s (a), 120 s (b) and 180 s (c), respectively details in the feeding area, in the same time intervals (a'-c')

Figure 9 shows the vector projection of the velocity at t=180 s for the clarifier with the supply pipe of 1m.

The vortex area can be easily observed, and further fluid propagation can be expected. Also, the tendency to move can be expected, at least for another period, in the bottom area of the decanter. This disadvantages the sedimentation process because the sediments are already on the bottom of the sedimentation tank and can be taken up by the water flow and lifted into the fluid mass. This will increase the final sedimentation time.

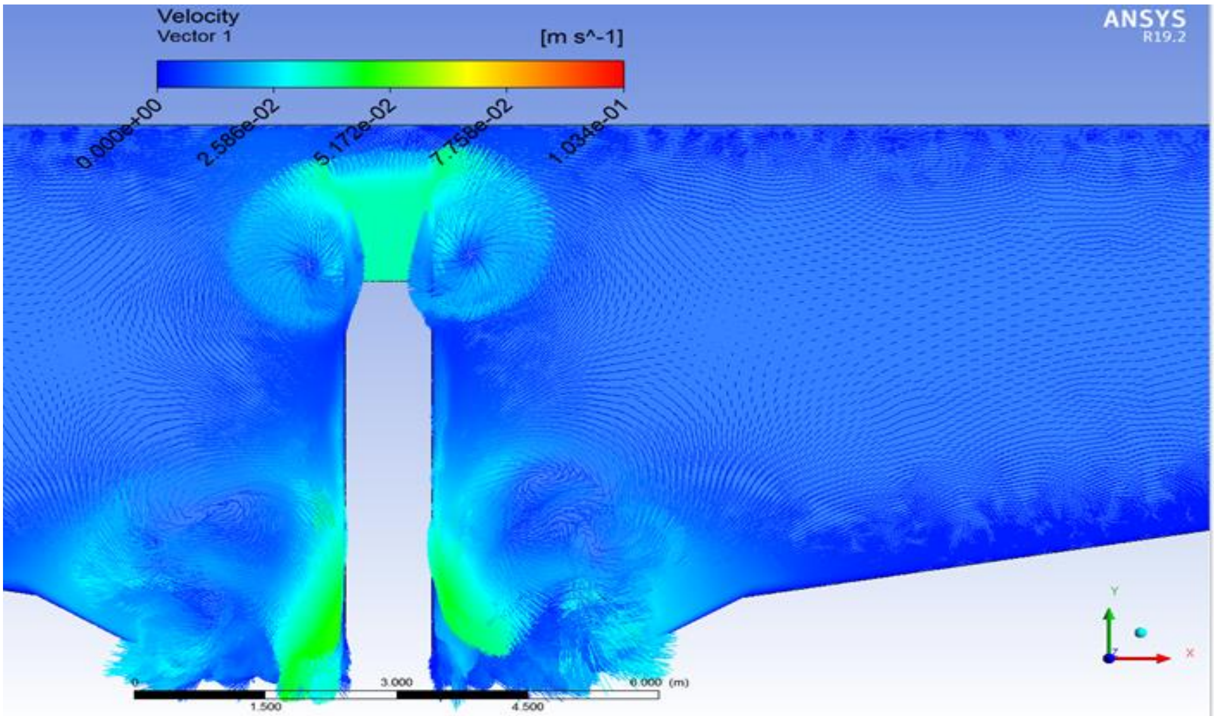


Fig. 9 - Vector projection of the velocity at t=180 s for the clarifier with 1m supply pipe

Reynolds number analysis

Figure 10 displays the Reynolds number's distribution along the diameter of the clarifier with the feeding pipe of 1 m, at 60 s, 120 s and 180 s. Also, details from the feeding zone, for the same time are presented here. Although for the supply area (inside the pipe) the calculated Reynolds number determined a turbulent regime, immediately after the entry of the fluid inside the clarifier, the regime becomes laminar, which favours the sedimentation process.

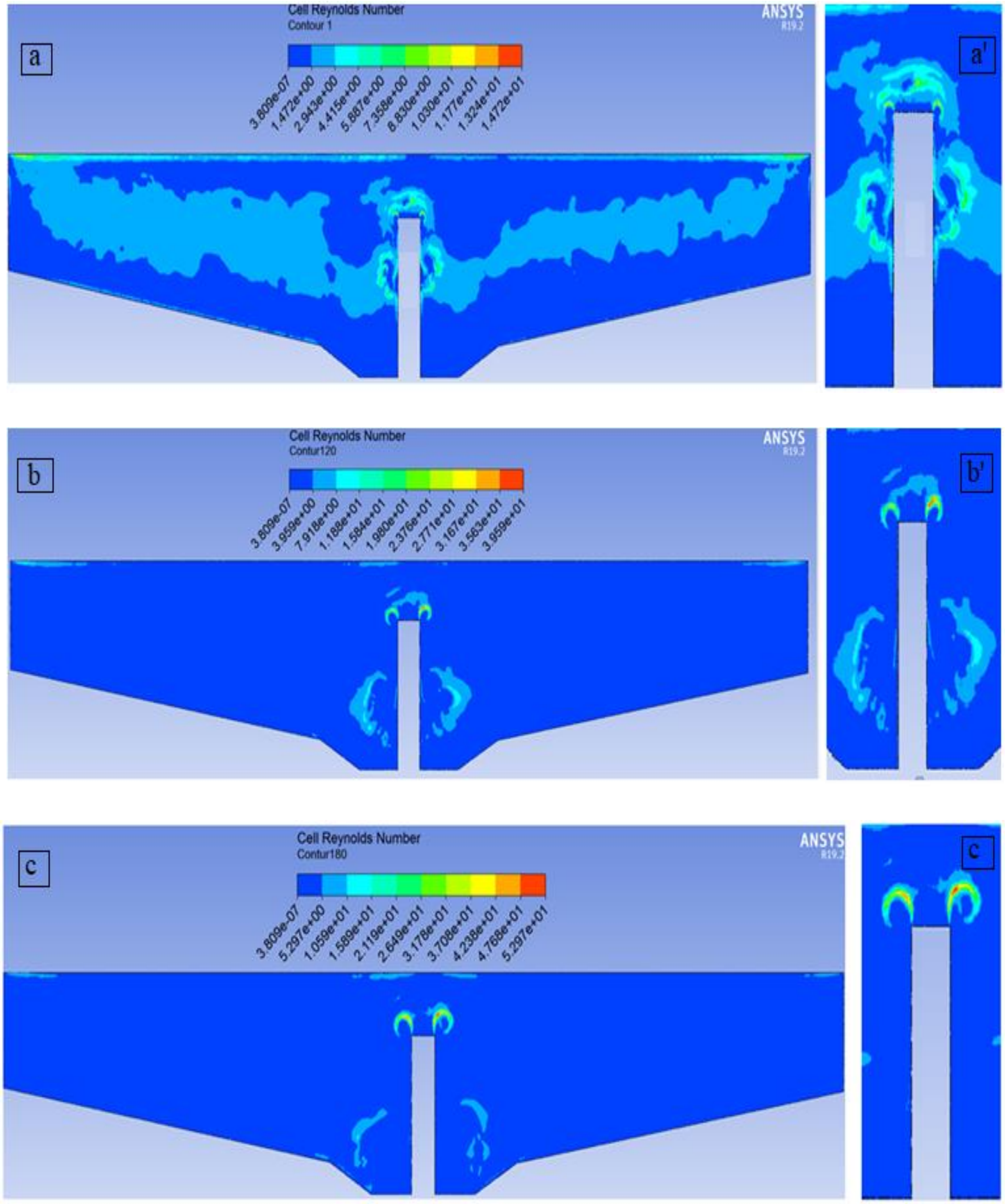


Fig. 10 - Reynolds number's distribution at 60 s (a), 120 s (b) and 180 s (c), respectively details of the feeding area, in the same time intervals (a'-c')

Turbulence intensity analysis

Figure 11 shows the distribution of turbulence intensity along the diameter of the clarifier with the feeding pipe of 1 m, at 60 s, 120 s and 180 s. The literature describes turbulence intensity as the ratio of standard deviation of fluctuating fluid velocity to the mean fluid speed, and it represents the intensity of fluid velocity fluctuation (*Zhang, 2013; Mudde et al., 2005)*.

If the maximum turbulence intensity remains constant at $3.82 \cdot 10^{-2}$ % over the entire length of the simulation time, it is interesting to analyse the evolution in time of the turbulence intensity over the length of the clarifier.

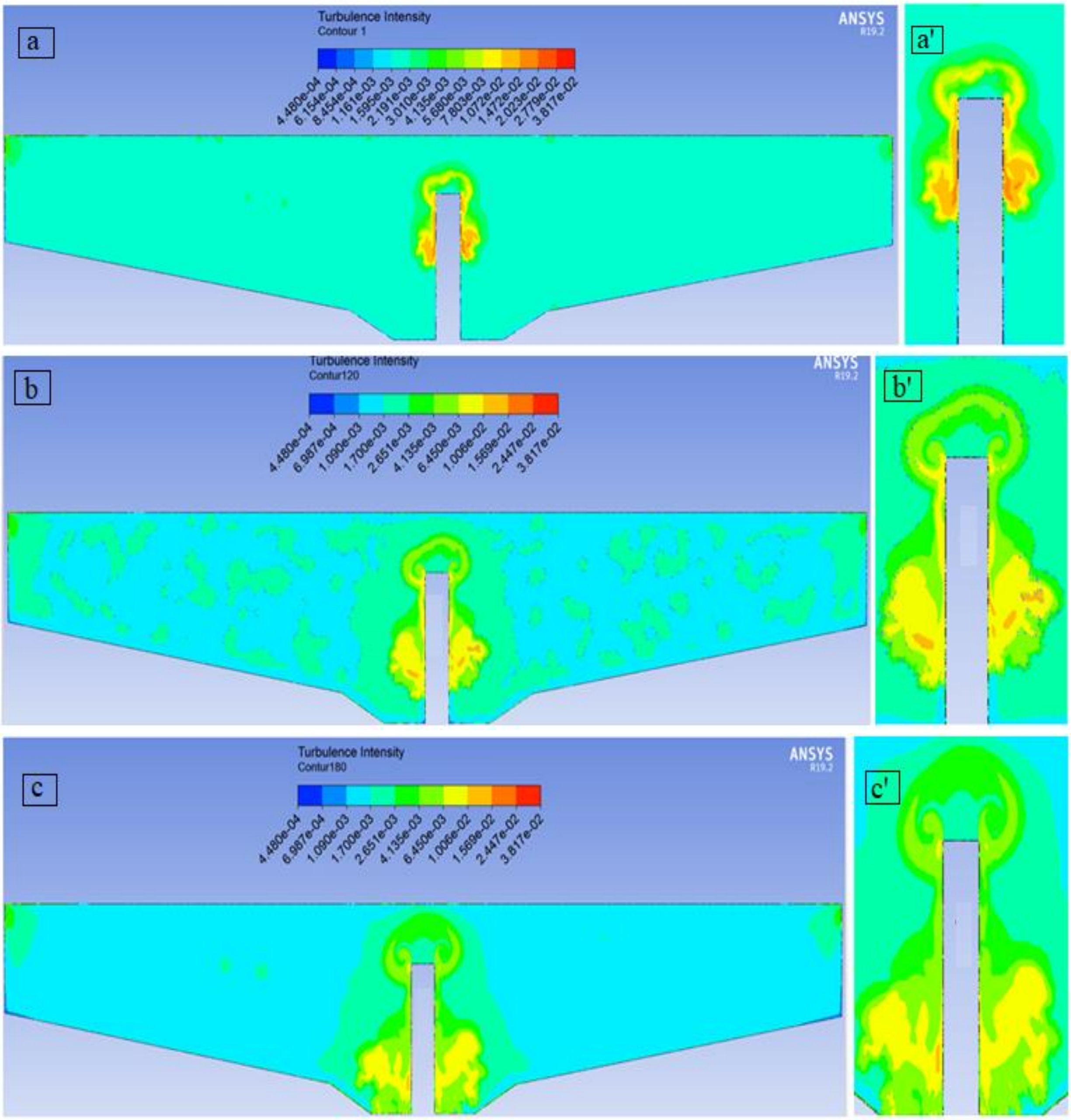


Fig. 11 - Clarifier turbulence distribution at 60 s (a), 120 s (b) and 180 s (c), respectively details of the feeding area, in the same time intervals (a'-c')

Analysing the results obtained for the clarifiers with 1 m supply pipe, regarding the fluid velocity analysis, despite the feeding area's velocity being set to 0.036 m/s, it is apparent from this number that there are locations in the clarifier where the velocity is 0.103 m/s. In the feeding area, velocity values are at their highest values. As presented in Figure 8, it is shown that the direction of travel is typically in the direction of the clarifier's bottom. This is caused by the vortices that form at the supply pipe's left and right corners at the end of the pipe. This results at the bottom of the clarifier in a considerable pressure difference in the fluid mass.

The Reynolds number study's highest values for the clarifier diameter were 14.7 for t=60 s, 39.6 for t = 120 s, and 53 for t=180 s. Although, initially, the tendency to move is towards the water evacuation areas, due to the appearance of vortices in the limit areas of the supply pipe, the fluid velocity increases, and proportionally with it, the Reynolds number also increases. Additionally, it could be seen that the presence of eddies affects the shift in travel direction from "mainly to the outlet" to "largely to the bottom" of the clarifier. Looking at the evolution of the Reynolds over time, an increasing tendency can be seen, but it will not increase so much as to go out of the area of a laminar regime on the diameter of the clarifier.

Also, regarding the turbulence intensity analysis showed that at t=60 s, the average value of the turbulence intensity was $1.70 \cdot 10^{-3}$ % but, with the appearance of vortices, which cause part of the fluid to move to the bottom of the clarifier, the turbulence intensity decreases in the radial direction of the clarifier to $1.09 \cdot 10^{-3}$ % at t=120 s and t=180 s. In time, the zones of maximum for the turbulence intensity are getting closer and closer to the bottom of the clarifier.

It should be mentioned that the results obtained agree with the results obtained by other researchers *(Chero et al., 2019; Griborio et al., 2021; Griborio et al., 2014; Shahrokhi et al., 2013; Czernek et al., 2014; Sharifi et al., 2019).*

CONCLUSIONS

Analysing the results obtained for the clarifiers with 1m supply pipe, regarding the fluid velocity analysis, despite the feeding area's velocity being set to 0.036 m/s, it is apparent from this number that there are locations in the clarifier where the velocity is 0.103 m/s. In the feeding area, velocity values are at their highest values.

In terms of the results of the Reynolds number analysis, the values increased during the three time periods looked at, with the minimum value being 14.7 and the maximum being 53.

Also, regarding the turbulence intensity analysis showed that at t=60 s, the average value of the turbulence intensity was $1.70 \cdot 10^{-3}$ % but, with the appearance of vortices, which cause part of the fluid to move to the bottom of the clarifier, the turbulence intensity decreases in the radial direction of the clarifier to $1.09 \cdot 10^{-3}$ % at t=120 s and t=180 s. The results obtained agree with the results obtained by other researchers. This work may help researchers in the field, mainly, but also the builders of radial clarifiers.

ACKNOWLEDGEMENT

This work has been funded by the European Social Fund from the Sectoral Operational Programme Human Capital 2014-2020, through the Financial Agreement with the title "Training of PhD students and postdoctoral researchers in order to acquire applied research skills - SMART", Contract no. 13530/16.06.2022 - SMIS code: 153734 and from the project "Development of the practical application base for agricultural, mechatronic and environmental mechanics in vineyards, orchards and solariums (DEMEVILISO)", CNFIS-FDI-2023-F-0277, from the Ministry of Education through the Executive Agency for Financing Higher Education, Research, Development and Innovation.

REFERENCES

[1] Alsina X. F. (2008), *Conceptual design of wastewater treatment plants using multiple objectives*. PhD Thesis, Faculty of Sciences at the University of Girona, Spain;

[2] Al-Jeebory A. A., Kris J., Ghawi A.H. (2010), Performance improvement of water treatment plants in Iraq by CFD model. *QJES*, *3*(1), 1-13;

[3] Brennan D. (2001), *The numerical simulation of two-phase flows in settling tanks.* PhD Thesis, University of London, London;

[4] Campbell B.K., Empie, H.J. (2006), Improving fluid flow in clarifiers using a highly porous media. *Journal of Environmental Engineering.*, 132, 1249-1254;

[5] Chero E., Torabi M., Zahabi H., Ghafoorisadatieh A., Bina, K. (2019), Numerical analysis of the circular settling tank Drinking Water Engineering and Science, *12*, 39–44;

[6] Czernek K., Ochowiak M., Janecki D., Zawilski T., Dudek L., Witczak S., Krupińska A., Matuszak M., Włodarczak S., Hyrycz M., Pavlenko I. (2014), Sedimentation Tanks for Treating Rainwater: CFD Simulations and PIV Experiments. *Energies, 14*, 7852. https://doi.org/10.3390/en14237852;

[7] Das S., Bai H., Wu C., Kao J.H., Barney B., Kidd M., Kuettel M. (2016), Improving the performance of industrial clarifiers using three-dimensional computational fluid dynamics. *Engineering applications of computational fluid mechanics*., 10, 130-144;

[8] Gernaey K.V., & Vanrolleghem P.A. (2005), Modeling of reactive primary clarifier. *Water Science and Technology*., 43(7), 73–81. doi:10.2166/wst.2001.0393;

[9] Griborio G., Rodríguez J.A., Enriquez L. (2021), Use of three-dimensional computational fluid dynamics model for a new configuration of circular primary settling tank McCorquodale. *Water Science and Technology*., *84* (2), 333–348;

[10] Griborio A., McCorquodale J.A., Rodriguez J.A. (2014), CFD Modeling of Primary Clarifiers: The State-of-the-Art. *In: Proceedings of the WEFTEC 87th Annual Technical Exhibition and Conference*. New Orleans, LA. doi:10.2175/193864714815941540;
[11] Hasim A.M.H., El-Hafiz A.A., El Baz A.R., Farghaly S.M. (2017), Study the performance of circular clarifier in existing potable water treatment plant by using computational fluid dynamics. *XVI World Water Congress*, Cancun;
[12] Hoffmann S., Feldmann U., Bach P.M., Binz C., Farrelly M., Frantzeskaki N.A. (2020), Research Agenda for the Future of Urban Water Management: Exploring the Potential of Nongrid, Small-Grid, and Hybrid Solutions. *Environmental Science & Technology*., *54*, 5312–5322. doi: 10.1021/acs.est.9b05222;
[13] Kohnke P. (1999), ANSYS Theory Reference. Eleventh Edition, SAS IP, Inc.;
[14] Mudde R.F., Deutz L., Nievaart V.A., van Maanen H.R.E. (2005), LDA-Measurements of the Turbulence in and Around a Venturi. *Proceedings of the ERCOFTAC International Symposium on Engineering Turbulence Modelling and Measurements*; ETMM6, Sardinia, Italy, 23–25 May, 511-520;
[15] Öberg G., Metson G.S., Kuwayama Y., Conrad S. (2020), Conventional Sewer Systems Are Too Time-Consuming, Costly and In-flexible to Meet the Challenges of the 21st century. *Sustainability, 12*, 6518. doi:10.3390/su12166518;
[16] Parry A. (2014), *Numerical Simulations and Optimisation of Gas-Solid-Liquid Separator. MSc Thesys in Petroleum Engineering,* Imperial College London, London;
[17] Patziger M., Günthert F.W., Jardin N., Kainz H., Londong J. (2016), On the design and operation of primary settling tanks in state-of-the-art wastewater treatment and water resources recovery. *Water Science and Technology., 74 (*9), 2060–2067. doi:10.2166/wst.2016.349;
[18] Rus F. (2001), *Separation operations in the food industry*, (*Operații de separare în industria alimentară*) Publisher: Transylvania University Publishing House, Romania;
[19] Saady N.M. (2012), Effect of inclined plates and polyelectrolyte on the performance of settling tanks. Department of Civil and Environmental Engineering, University of Windsor;
[20] Sharifi Kh., Jafari B.T., Ebrahimi S., Sabeti M., Soflaee S. (2019), A new computational fluid dynamics study of a liquid-liquid hydrocyclone in the two phase case for separation of oil droplets and water*, Brazilian Journal of Chemical Engineering 36*, 1601-1612;
[21] Shahrokhi M., Rostami F.Md., Said M.A., Sabbagh Yazdi S.R., Syafalni S. (2013), Computational investigations of baffle configuration effects on the performance of primary sedimentation tanks. *Water and Environment Journal., 27* (4), 484–494. doi:10.1111/j.1747-6593.2012.00367.x;
[22] Smol M. (2022), Chapter 1 - Circular economy approach in the water and wastewater sector, Editor(s): Alexandros Stefankis, Ioannis Nikolaou, Circular Economy and Sustainability, *Elsevier*, 1-19, ISBN 9780128216644.;
[23] Smol M., Koneczna R. (2021), Economic Indicators in Water and Wastewater Sector Contributing to a Circular Economy (CE). *Resources, 10*, 129. https://doi.org/10.3390/resources10120129;
[24] Yakhot V., Orszag S.A., Thangam S. (1992), Development of turbulence models for shear flows by a double expansion technique. *Physics of fluids. 4*, 1510-1520;
[25] Zăbavă B.Ș., Constantin G.A., Voicu G. (2021), Numerical CFD analysis in a radial decanter. *UPB Scientific Bulletin* Series D, 83 (3).***;***
[26] Zhang L.Z. (2013), Conjugate heat and mass transfer in heat mass exchanger ducts, 1st ed.; Publisher: Academic Press;
[27] ***Analysis Tools, (2008), ANSYS Advantage, Volume 2, no. 4, https://www.ansys.com/content/dam/company/advantage/aa-v2-i4-full-version.pdf ;
[28] ***Clarifier Design: WEF Manual of Practice No. FD-8, Water Environment Federation. (2005), 2nd ed McGraw-Hill Publishing House. https://silo.pub/clarifier-design-wef-manual-of-practice-no-fd-8.html ;
[29] ***Stat Trek, Residual Analysis in Regression. https://stattrek.com/regression/res idual-analysis.aspx;
[30] ***Manuals of British Practice in Water Pollution Control. PRIMARY SEDIMENTATION, (1980), The Institute of Water Pollution Control https://www.ircwash.org/sites/default/files/341.9-80UN-1158.pdf.