Research, Society and Development, v. 9, n. 4, e180942542, 2020
(CC BY 4.0) | ISSN 2525-3409 | DOI: http://dx.doi.org/10.33448/rsd-v9i4.2542

Simulação de um tanque industrial de tratamento de resíduos

Simulation of un industrial waste treatment tank

Simulación de un tanque de tratamiento de residuos industriales

Recebido: 10/01/2019 | Revisado: 11/02/2020 | Aceito: 17/03/2020 | Publicado: 20/03/2020

Elcio Fernando dos Santos Pereira
ORCID: https://orcid.org/0000-0003-1534-0617
Universidade São Paulo, Brasil
E-mail: elcio.pereira@gmail.com
Luiz Mário Nelson de Góis
ORCID https://orcid.org/0000-0001-8536-8936
Universidade Salvador, Brasil
E-mail: luiz.gois@unifacs.br

Resumo

O objetivo do presente trabalho foi avaliar a operação de um tanque de sedimentação industrial utilizado na separação de resíduos sólidos da indústria petroquímica. Os dados de profundidade foram obtidos através de um "flutuador de interface", enquanto os diâmetros e as posições das partículas passaram pela simulação de CFD. O simulador computacional de dinâmica de fluidos (FLUENT 6.3.26) foi utilizado para realizar uma simulação multifásica utilizando a abordagem de Euler-Lagrange e foi utilizado para determinar as trajetórias de partículas e cotores de sólidos acumulados no fundo do tanque. Isso permitiu uma melhor compreensão do acúmulo de sólidos e melhoria do processo de limpeza. Na simulação do tanque, uma grande malha computacional compreendendo 464.094 nós computacionais foi projetada. O uso da abordagem de Euler-Lagrange significou que um modelo de fase discreta teve que ser estabelecido e os parâmetros do modelo de distribuição de sólidos de Rosin-Rammler para as condições de contorno da simulação tiveram que ser determinados.

Palavras-chave: Tanque decantador; simulação, CFD

Abstract

The objective of the present work was to evaluate the operation of an industrial sedimentation tank used in the separation of solid waste from the petrochemical industry. The depth data were obtained through a “interface float”, while the diameters and the positions of the particles through the CFD simulation. The computational fluid dynamics simulator (FLUENT 6.3.26) was used to perform a multiphase simulation using the Euler-Lagrange approach and was used to determine the particles trajectories and cotours of solids accumulated in the bottom of the tank. This allowed a better understanding of solids accumulation and improvement of the cleaning process. In the simulation of the tank a large computational mesh comprising 464,094 computational nodes was designed. The use of the Euler-Lagrange approach meant that a discrete phase model had to be established and the parameters of Rosin-Rammler solids distribtion model for the boundary conditions of the simulation had to be determined.

Keywords: CFD, settling tank; simulation

Resumen

El objetivo del presente trabajo fue evaluar la operación de un tanque de sedimentación industrial utilizado en la separación de residuos sólidos de la industria petroquímica. Los datos de profundidad se obtuvieron a través de una "interfaz flotante", mientras que los diámetros y las posiciones de las partículas pasaron a través de la simulación CFD. El simulador de computadora de dinámica de fluidos (FLUENTE 6.3.26) se usó para realizar una simulación multifase usando el enfoque de Euler-Lagrange y se usó para determinar las trayectorias de partículas y sólidos acumulados en el fondo del tanque. Esto permitió una mejor comprensión de la acumulación de sólidos y la mejora del proceso de limpieza. En la simulación del tanque, se diseñó una gran malla computacional que comprende 464,094 nodos computacionales. El uso del enfoque de Euler-Lagrange significó que se debía establecer un modelo de fase discreta y se debían determinar los parámetros del modelo de distribución de sólidos de Rosin-Rammler para las condiciones de contorno de la simulación..

Palabras clave: decantador; simulación,;, CFD

1. Introduction

Recently studies on settling phenomena have mostly focused on evaluating the effects of the gravitational field on particles dispersed in a suspension. However studies on rheological effects and influence of porous media or even geometrical concerns are complementing the understanding of this multiphase problem. The use of more and more powerful computers has helped in the application of numerical methods and the use of combination of experiments and simulations is yelding better results.

The settling tank studied in this work with is a unit installed in a wastewater treatment plant of a petrochemical company in Candeias, Bahia, Brazil. In Figure 1 an aerial photo of the tank is shown. Its dimensions are 98.0m long, 38.0 wide and 2,95m in height which gives a volume of 10,985.80 m^3. Due to its large dimensions and the volume of waste that is treated in the tank, it is possible to perceive its importance in the plant's waste treatment unit.

Figure 1- Aerial photograph of the tank (S*ource:* ***Dow Química do Brasil***)

In Figure 2, the tank feed is shown. It is made through a 1.0m diameter tube that discharges the liquid mixture from a height of up to 2,95m and at a flow rate of 1,344.94 ton/h.

Figure 2- Inlet pipe feeding the tank (*source: Dow Química do Brasil*)

Figure 3 shows the detail of the spillway from the tank where clear liquid is released. The spillway is located on the right margin of the tank outlet and has a length of 4,0m and a height of 0,5m, forming an area of 2,0m^2.

Figure 3- Clearer water in the outflow **(Source: *Dow Química of Brasil*)**

In the literature, some authors have applied coputational fluid dynamics (CFD) to improve knowledge of settling phenomena and their results have been used in the design and evaluation of settlers. Authors like Matko *et al* (1996), Kahane *et al* (2002), Ekama *et al* (2004), Flamant (2004), Dickenson and Sansalone (2009), Rostami *et al* (2011), Yu *et al* (2013), Ramin *et al* (2014), performed stuies by applying new and comprehensive models to explain how settlers and solid separators work. More recently, two works presented applications of CFD to treatment plants. Yu *et al* (2013), who applied CFD to a biodigester used for the generation of methane and Ramin *et al* (2014) who proposed a new model to explain to setting speed in an activated sludge tank.

Therefore, the present work was done with data obtained from a large industrial tank, different from the laboratories or pilot units data like obtained by other authors.

2. Methods

The present work consists of an exploratory study related to the evaluation of the operation of a large industrial tank. the investigation was done based on information concerning the data of the depth of the depth and the position of the solid deposit formed at the base of the tank. The methodology used were two, one for the data of depth of the solid layer and the other for the data of concentration of solids in different positions of the tank. For the depth data of the solid an immersion buoy was used that was introduced in the tank and the height of the surface was measured until the interface where the solid deposit was located. This methodology is shown through Figure 4, where the drawing of an operator can be seen inserting the immersion float up to the interface of the solid tank. For the concentration data, positions in the length of the tank were chosen and samples were collected.

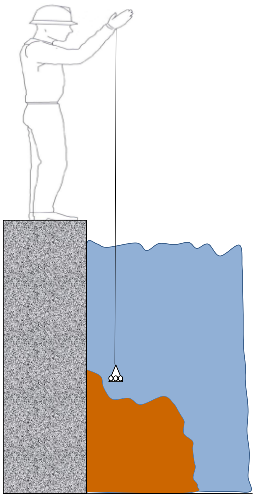

Figure 4 – Measurement of solid-liquid interface. (*Source: the author*)

The experimental part of this work was performed by the height of the interface between accumulated solids and the liquid along the tank. For this task an "interface float" was used as shown in Figure 4. Measurements were taken when solids were observed through the interface float.

The concentration data were taken at 10 different positions along the tank (Table 1).

Table 1- Positions for taking samples.

Position	**x (m)**
1	37.0
2	48.0
3	54.0
4	60.0
5	66.0
6	72.0
7	78.0
8	84.0
9	90.0
10	96.0

(Source: the Author)

(CC BY 4.0) | ISSN 2525-3409 | DOI: http://dx.doi.org/10.33448/rsd-v9i4.2542

The lengths indicated in Table 1 for each position were the distances measured in relation to the tank edge.

3. The CFD Simulation

A simulation of the settling tank was performed by applying the steady, state finite volume method and the Euler approach. Particle size distribution was determined using the Rosin-Rammler model [Crowe *et al* (1998)] According to Veselind (1980) this model is suitable for application to tanks used for the treatment of solids in wastewater systems.

The feed stream was a mixture of solids dissolved in water, with a flow rate of 1,308 m^3/h. Calciun cabonate was the most representative product accouning for 28.53% of the total solids present in the feed stream, and hence the feed stream was considered to consist of just one solid.

A numerical mesh was designed using hex/wedge elements with a node spacing of 0,3m resulting in a total of 464,094 numerical nodes in the tank. Figure 5 shows the numerical network in further details.

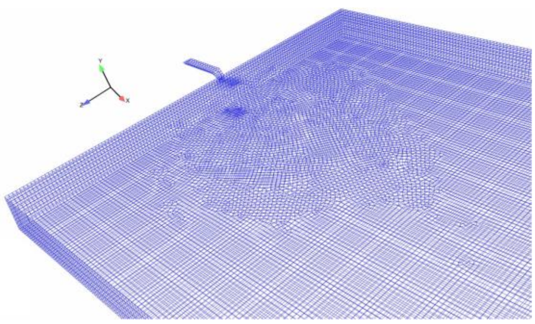

Figure 5- Numerical mesh.*(Source: the author)*

(CC BY 4.0) | ISSN 2525-3409 | DOI: http://dx.doi.org/10.33448/rsd-v9i4.2542

In the detail of Figure 5, it is possible to observe the large number of finite elements and nodes that formed the representative mesh of the tank, which resulted in 43,000 iterations so that convergence in the simulation could be obtained

3. Results and discussion

One of the results of this work was a solid-liquid interface height profile in several points of the length of the tank. Figure 6 shows the reduction in the height (h) of the solid-liquid interface as a function of the distance (x) from the feed pipe.

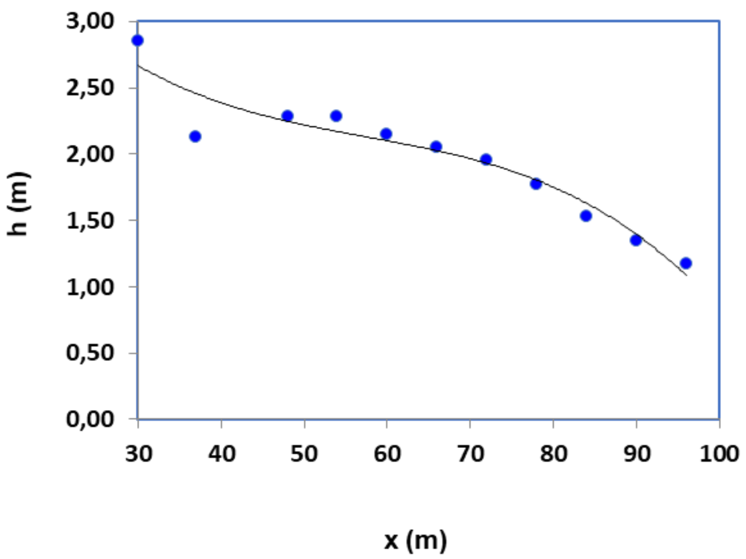

Figure 6: Solid interface height as a function of distance. *(Source: the author)*

Figure 6 shows a reduction in the height of the interface with the length of the tank. That is, the more distant the samples were collected, the lower the interface heights with the solid phase. This result was expected because, with a reduction in sedimentation velocity, particles tend to accumulate near the feed tube**.**

Figure 7 shows a case in which there was an excess of solids accumulated in the tank

Figure 7 – Solids accumulation in the tank.*(Source: the author)*

As seen in Figure 7, the accumulated solids filled the entire tank, mainly in the vicinity of its feed. This situation occurred systematically in the unit, causing the operation of the tank to be interrupted.

3.1 Simulation results

The CFD simulation results were observed as histograms of size distribuition vs concentration of particles at the positions listed in Table 1. Figures 8 to 12 show five histograms corresponding to positions 1, 3, 5, 7 and 9, of Table 1.

Figure 8 shows the histogram of the concentration of particles in position 1 of the tank or up to 37m from the tank supply.

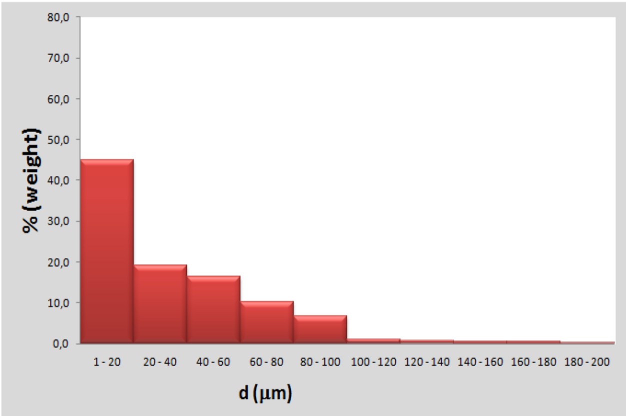

Figure 8: Particle size distribution at position 1 (Source: the author)

As seen in Figure 8, there is a large concentration of particles with diameters up to 20mm in the position 1 of the tank.

Figure 9 shows the concentration histrogram for position 3 or at 54m from the tank supply.

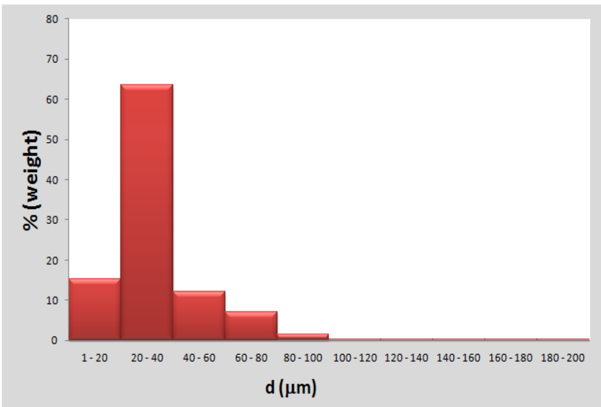

Figure 9: Particle size distribution at position 3. *(Source: the author)*

As seen in Figure 9 there is a large concentration of particles with diameters between 20 and 40 mm in position 3 of the tank.

Figure 10 shows a histogram of concentrations for position 5, that is, at 66m from the tank supply.

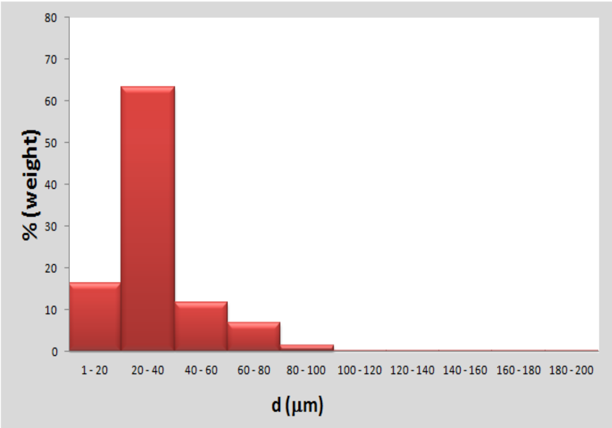

Figure 10: Particle size distribution at position 5. *(Source: the author)*

Figure 10 also shows a large concentration of particles with diameters between 20 and 40 mm in position 5 of the tank.

Figure 11 shows a histogram of concentrations for position 7, at 78m from the tank supply.

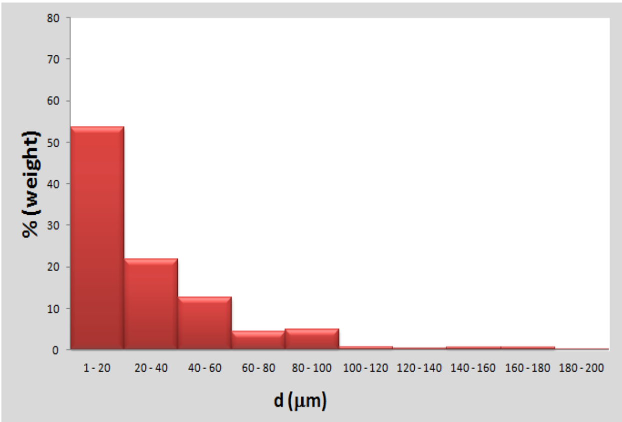

Figure 11 Particle size distribution in the position 7. *(Source: the author)*

As seen in Figure 11, there is a large concentration of particles with diameters up to 20 mm in position 7.

Figure 12 shows a histogram of concentrations for position 9, that is, at a distance of 90m from the tank supply.

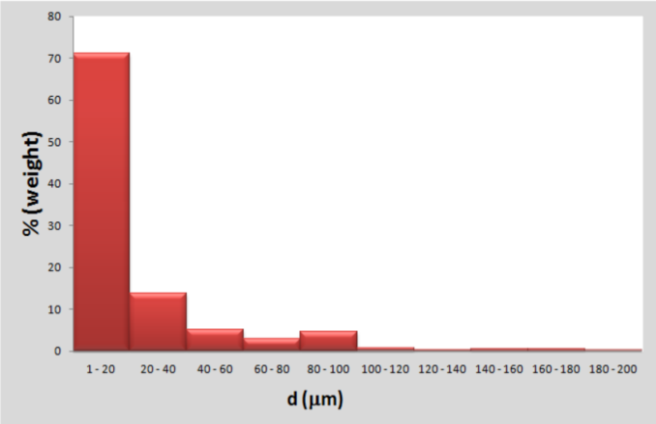

Figure 12: Particle size distribution at position 9 *(Source: the author)*

As seen in Figure 12, there is a large concentration of particles with diameters up to 20µm in position 9 of the tank

Analyzing the graphs in figures 8 to 12, the histograms indicate different behaviors for particles with diameters from 1 to 20 µm and from 20 to 40 µm. Particles in the range of 1 to 20 µm show a decrease in concentration and weight up to a distance of 66m from the feed (position 5). After this point their concentration and weight increase. On the other hand, the concentration and weight of particles in the range of 20 to 40 µm increase in the samples taken up to 54 m (position 3) and then decrease up to distance of 66 m from the feed pipe; thereafter, their concentration remains constant.

This particles behavior observed along the tank is related to what is observed in the first histogram and can be justified as follows. As the stream is fed into the tank, the larger and heavier particles reach the bottom of the tank are dragged back to the flowing stream and settler toward the front of the board a new position forward in the tank. This occurs within a distance of 50 m from the board (position 5) and ceases when the gravitational field finally stops the particles flow near to the outflow.

The particles (1 to 20 µm), display a different behavior. Owing to their small diameters, the particles have a low terminal settling velocity and they reach the bottom of the tank further than 50 m from the feed pipe. The position of the outflow makes it easier for the lighter particles to move forward in the last 60 m through the action of the drag force.

4. Final considerations

The obtained results indicate that solids tend to accumulate the region close to the feed pipe, i.e., within 50 m from the pipe. This accumulation is mainly attributed to heavy particles having diameters greater than 20 μm. In the region from 70 to 100 μm from pipe, larges particles with diameters from 1 to 20 μm are concentrated.

Comparing Figures 6 and 7 and the histograms shown in figures 8 to 10 with Figure 4, it is easy to conclude that particles having diameters from 20 to 40 μm accumulate in the close to the board of the tank.

The CFD simulation results obtained herein can significantly aidsaid petrochemical companies in planning and performing regular cleaning and ensuring normal operation of settling tanks.

Thus, the great merit of the research in the present work is that with the data obtained, both from the simulation and from the heights of deposits, a prediction of the deposition of particles in the tank has been achieved. This forecast involves the diameter of the deposited particles, their position in relation to the feed and the height of the deposit formed. These data are of great importance for the operation of the tank since with them it is possible to predict a better operational situation, which has even brought financial profits to the company.

The main suggestion for future work is to study the trajectories formed by the particles present in the tank that can be obtained from CFD simulations. This study should complement the present research, contributing even more in the evaluation of the operation of the settling tank evaluated here.

REFERENCES

Crowe, C.T., Sommerfeld, M. & Tushi, Y. (1998). *Multiphase Flows with Droplets and Particles*. CRC Press: Boca Raton, FL.

Dickenson, J.A. & Sansalone, J. J. (2009). Discrete Phase Model Representation of Particulate Matter (PM) for Simulating Separation by Hydrodynamic Unit Operations. *Env. Sci.*, 43(1): 8220-8226.

Ekama, G. A. & Marais, P. (2004). Assessing the applicability of the 1D flux theory to full-scale secondary settling tank design with a 2D hydrodynamic model. *Water Research,* 38(3): 495-506.

Flamant, O. (2004). Experimental Analysis and Simulation of Settling Process. *Proc. Saf. Env. Prot.,*. 82(4): 312-318.

Kahane, R., Nguyen, T. & Schwarz, M. P., (2002). CFD modelling of thickeners at Worsley Alumina *App. Math. Mod.,* 26 (02): 281-296.

Matko, T. (1996). Recent Progress in the Numerical Modelling of Wastewater Sedimentation Tanks, *Proc. Saf. Env. Prot.,* 74(4): 245-258.

Ramin, E., (2014). A new settling velocity model to describe secondary sedimentation, *Water Research*, 66: 447-458.

Rostami, F. (2011). Numerical modeling on inlet aperture effects on flow pattern in primary settling tanks. .*Appl. Math. Mod*.35(6): 3012-3020.

Veselind, P. A. (1980). The Rosin-Rammler Particle size distribution. *Resource Recovery and Conservation,*. 5: 275 – 277.

Yu, L. (2013). Multiphase modeling of settling and suspension in anaerobic digester. *Appl. Energy*, 111: 28-39.

Porcentagem de contribuição de cada autor no manuscrito

Elcio Fernando dos Santos Pereira - 50%

Luiz Mário Nelson de Góis - 50%