EFFICIENCY COMPARATION OF TANKS OF PRE-FLOTATION USING COMPUTATIONAL FLUID DYNAMICS (CFD) SIMULATION

Fábio Pereira dos Santos, fsantos@esss.com.br
Carlos Eduardo Fontes, carlos.fontes@esss.com.br
João Américo Aguirre Oliveira Jr., aguirre@esss.com.br
Engineering Simulation and Scientific Software Ltda.

Marcel Vasconcelos Melo, mvmelo@petrobras.com.br
Petroleo Brasileiro S.A.

Abstract. *In mature onshore fields, the volumes of oily water (water + oil) that are produced in petroleum production have been increasing. So, we have been concerned what one should do with this water. In this scenario, the reinjection of water in oil wells has been done with more and more frequency .However, this procedure imposes some cares with the geological formation to avoid damages that could cause reduction in oil production. The reinjection quality relies on the treatment of the water. If these treatments are deficient, the water reinjection could promote the formation of insoluble composts inside of the reservoir. In this context, we decided to evaluate alternative technologies or modified existent technologies that would be capable to increase the capacity and separation efficiency. We have used the Computational Fluid dynamic (CFD) to simulate three different separation tanks configurations (differing in internal equipment configuration) to estimate the separation efficiency of each of these models. The tank with the highest efficiency value could be used to increase the water injection in wells. The first configuration was compound of a simple tank with no internal devices; the second one presents a mixture distributor and two collectors, one for the water and other for the oil stream and; the third one with internal walls - chicanes - that would increase the total residence time of the mixture. It is important to observe that all cases have the same external dimension. Nine operational conditions were simulated for each geometry arrangement. Each one of these different conditions is obtained by varying mass flow rates oil concentration (totalizing 27 simulations). Our main objective was to verify the influence of these operational parameters in global efficiency. To obtain the results we have proposed a new methodology to simulate the internal devices apart from the main tank body and then we have connected this simulation result with the tank body using sources terms to represent the collectors and the distributor. As a result, the methodology that was proposed behaved satisfactorily, reducing the computational cost and aiding the engineers to choose the best configuration that enable to increase the process capacity.*

Keywords: *Tanks separators, Multiphase flow, Computational Fluid Dynamics (CFD) , Numerical simulation.*

1. INTRODUCTION

In mature onshore fields, the volumes of oily water (water + oil) that are produced in petroleum production increases continuously. So, engineers have been concerned about what they should do with this produced water. In this scenario the water reinjection has been a common practice and this process has been characterized as a profitable procedure. However, the reinjection of produced water imposes some special needs. For example, the oil content on the injected water must respect some maximum value defined in environmental codes and, also, in an ideal scenario all the oil in produced water is separated.

In this context, we decided to evaluate alternative technologies or modify existent technologies that would be able to increase the capability and separation eficiency of gravitational separators and flotator tanks. We used Computational Fluid Dynamics (CFD) to simulate three diferent configurations of tanks for oil/water separation (these tanks differ in internal devices configuration). Our goal is to define which one of those tanks should be used to increase the water injection in the reservoirs. The oil-water separator is a gravity separation device that is designed to increase the oil to be separated, based on density and on droplets diameter. Stamou(2008) has proved that simple modifications in internals configurations could increase significantly the tank efficiency of separation. To evaluate the influence of internals in the separator's efficiency, the modifications have been examinated by a CFD model.

1.1 Problem description

In this work we have evaluated three different configurations of a base tank used for flotation. One of them without any internal devices and the other two are with some internal devices (collectors and distributors or chicanes). Fig. 1 and Fig. 2 show the tanks that were studied and Tab. 1 and Tab. 2 show the internal devices data.

Recent work (Simmons et al, 2004) have studied gas-liquid-liquid separation and verified that resident time of gravitationals separators reduces if some baffle were included . The tank without internals and with internals devices (collectors and distributor) have been used by Petrobras.In this work a configuration with chicanes was chosen because this configuration has already been reported in the literature with significant increase in efficiency.

Table 1. Oil properties

Oil properties	Oil of "Canto do Amaro"
Viscosity cP	40
API	28

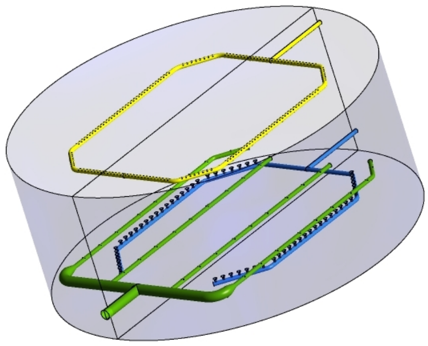

Figure 1. Tank with tubes internals devices - in blue the collector of water - in yellow the collector of oil - in green the ditribuitor of oily water

Table 2. Data of the tanks

Tube	Elevation, mm
Distruitor	1350
Water collector	710
Oil collector	8241

In all cases, the geometries had the same external dimension. They were represented by a cylinder with 22 meters in diameter and 9 meters in height (level of water). For this work, we have tested nine simulations for each geometric configuration, totalizing 27 simulation, as will be described in the table below:

Table 3. Operation Conditions

Total flow rate, m^3/d	Oil concentration, ppm		
36000	500	1500	2500
67500	500	1500	2500
90000	500	1500	2500

2. PHYSICAL MODEL

To measure the efficiency of these three tanks, we have used ANSYS CFX v11.0 to solve the conservation equations for the main water flow (Eulerian phase). To represent the droples of oil we have used the Lagrangian approach.Lagrangian approach should not be used whenever the particle volume fraction exceeds 0.12 ANSYS(2007). The two-way Lagrangian approach was applied because the oil concentration in the water was high enough to have some influence on the main water flow. The Lagrangian models solves a separate equation of motion (Newton's second law) for individual parcels (groups of particles).

The Lagrangian solution method integrates the three-dimensional trajectories of the parcels based on the forces acting on them from the surrounding fluid and other sources (i.e. body forces) Crowe(1993). Turbulence is usually accounted for by adding a random velocity component, superimposed over the average Eulerian velocity solved Bird, Stewart, W.E. and Lightfoot (2004). We have also considered the virtual mass force, this force is caused by the fact that the particle has to accelerate some portion of the surrounding fluid, leading to an additional drag. This approach was also possible because of the operation condition of the tank, that works in a semi-batch process. In this semi-batch operation, when the oil level reaches the oil outlet piping it is extracted. So we could use the hypothesis that the oil droplets were collected once they reach the water/oil interface (defined as the top of the simulation domain).

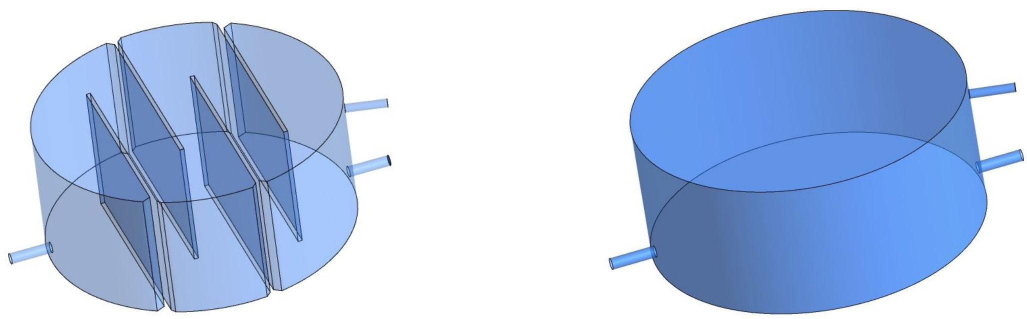

Figure 2. Tank with chicanes and without internals

2.1 Balance equations

As previously commented, in this work, Euler-Lagrange (in a two-way coupled fashion) simulations were performed, where water was the continuous phase and the oil was solved by a particle transport model. The equations below represent the mathematical model applied:

$$\nabla \cdot u = 0 \tag{1}$$

$$\rho(\frac{du}{dt} + u \cdot \nabla u) = -\nabla p + \mu \nabla^2 u + f \tag{2}$$

where, u represents velocity , p pressure , ρ density and f body forces. For the continuous phase, and:

$$m_p \frac{dU_p}{dt} = F_D + F_{VM} + F_t \tag{3}$$

The droplets movement equation. Where, m_p represents the mass of the droplet, U_p droplet velocity , F_D drag force, F_{VM} virtual mass force and F_t turbulent dispersion force (Crowe, Sommerfeld and Tsuji,1998).

2.2 Methodology

In the tank configurations without any internals and with chicanes, simulation domain represented the whole geometries, as can be seen in Fig. 2 . However, for the tank configuration with internal distributor and collectors, the simulation was divided in two steps: First, we simulated the internal equipments (distributor and collectors) for each operational condition, from that result, the velocity profile and mass flow rate in all points of injection and collection were extracted.

In the tank simulation, the distributor and collectors geometry was simplified into a large array of source points. So we have then fed the mass flow rate as an input for each injection and collection point along the domain using the data from the equipment simulation. This approach was applied to avoid the large computational cost of modeling all the pipings and distributor/collectors "holes" with a very fine mesh.

2.3 Sources points study

Prior to using the source points simplification, a study of the influence of this assumption was performed. In a very simple "quasi-two-dimensional" rectangular domain two cases were defined. In the first one, the small fluid injection was represented by a very fine mesh while; in the second one, a coarser mesh was used applying a mass and momentum source point to represent the boundary condition.

The results of these cases are compared in Fig. 4. As we can see, the use of source point in instead of using the fine mesh with the real boundary condition is the same that we expected when we use a coarse mesh on a typical CFD study. on a typical CFD study. No physical phenomena was lost by the application of the source point. Based on this study all the distributor/collectors piping was replaced in the simulations by arrays of source points (acting as sources for the distributor and as sinks for the collectors).

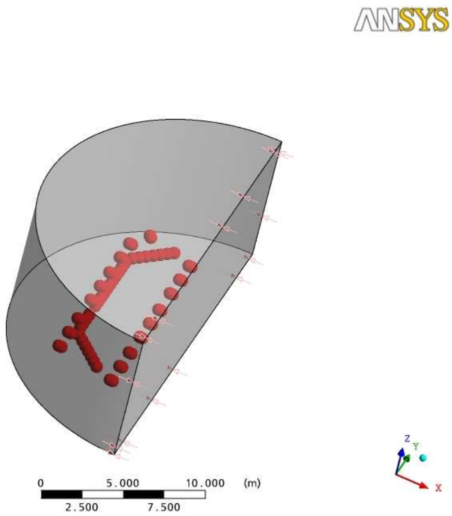

Figure 3. Tank internal distributor and collectors (points of injection in red).

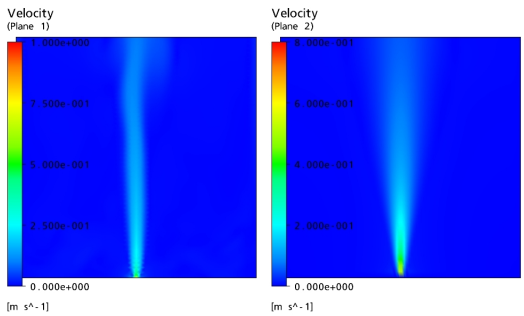

Figure 4. right side the real boundary condition and the other side a source term boundary condition.

2.4 Distributor and collectors modeling

As explained before, the solution of the flow inside the distributor and collectors used in the third separator configuration was done prior to the tank simulation itself. Figure 5 shows, in detail, the mixture distributor and the water collector. These simulation were performed by imposing the flow rate (bulk) at the end of the distributor/collectors piping. As one can see in Fig. 6, the distribuition of velocity along the distributor holes was nearly uniform while, in the collector, the flow rate distribution was not evenly distributed.

From the simulation, we can observe that not all the collection points are being used over the collectors piping, this may damage the operation of the tank by generating preferential paths and recirculation zones, reducind the overall residence time.

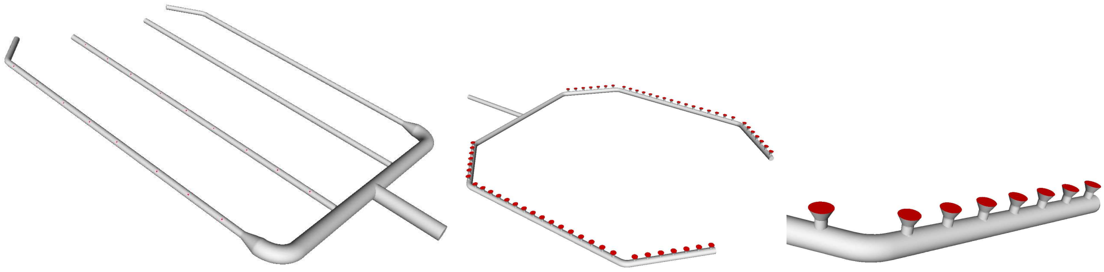

Figure 5. Internal devices, distributor and water collector, details.

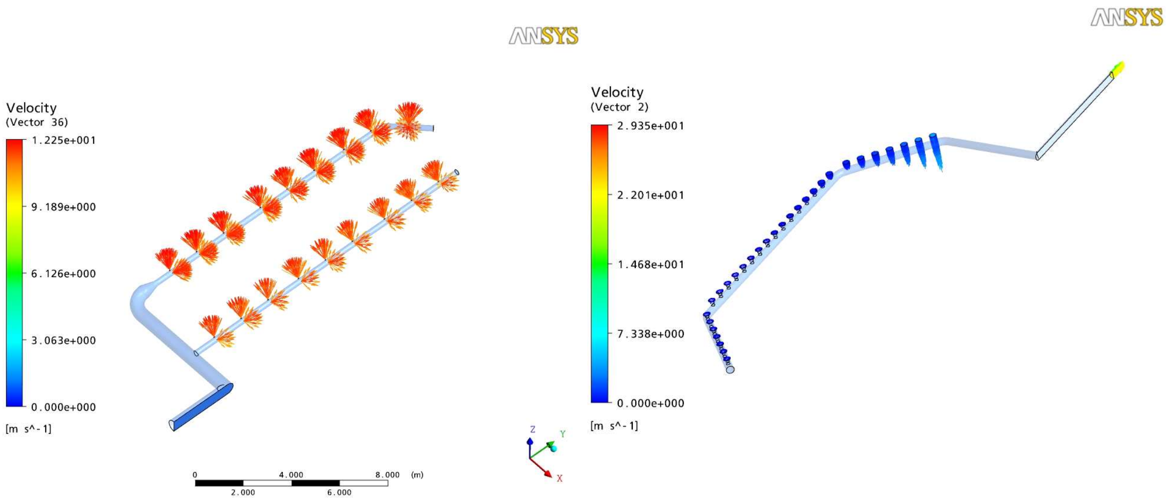

Figure 6. Velocity profile in distributor and in collector.

3. SIMULATION RESULTS

To calculate the efficiency of separation of the tanks, the oily water was supplied according to the distribution of droplets illustrated in Fig. 7, almost uniformly, for all tanks evaluated. Thus, we can infer the efficiency of separation in graphics below.

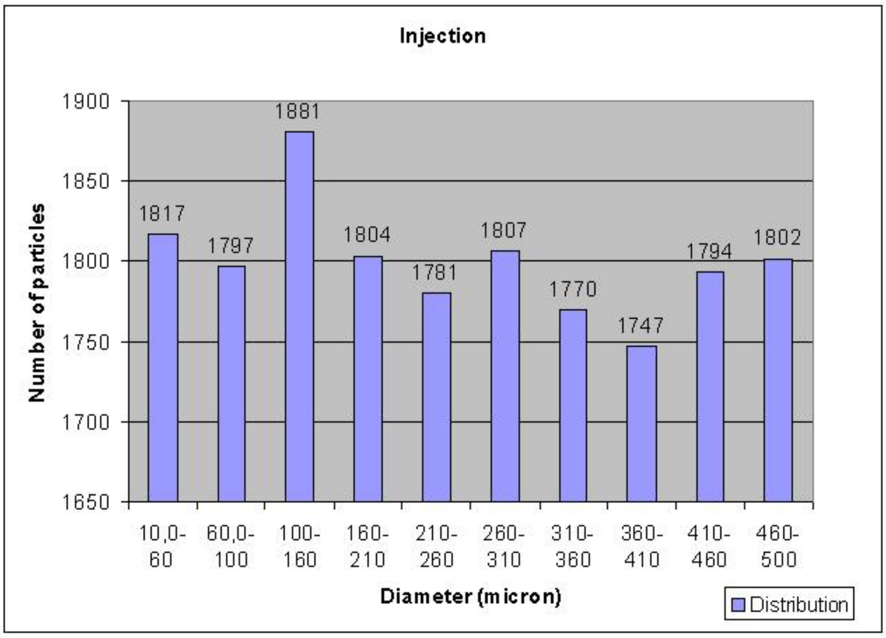

Figure 7. Droplets diameter distribuition in injection points.

Figure 8, Fig. 9 and Fig. 10 show the overall efficiencies of separation for the concentration oil of 500, 1500 and 2500 ppm, respectively. As expected, the tank equipped with chicanes is the one with the highest efficiency. The following equation represents the efficiency definition, $\eta = \frac{Nc}{Ne}$, where Nc is the number of particles collected in the top of the tank and Ne is the number of droplet injected in the tank;

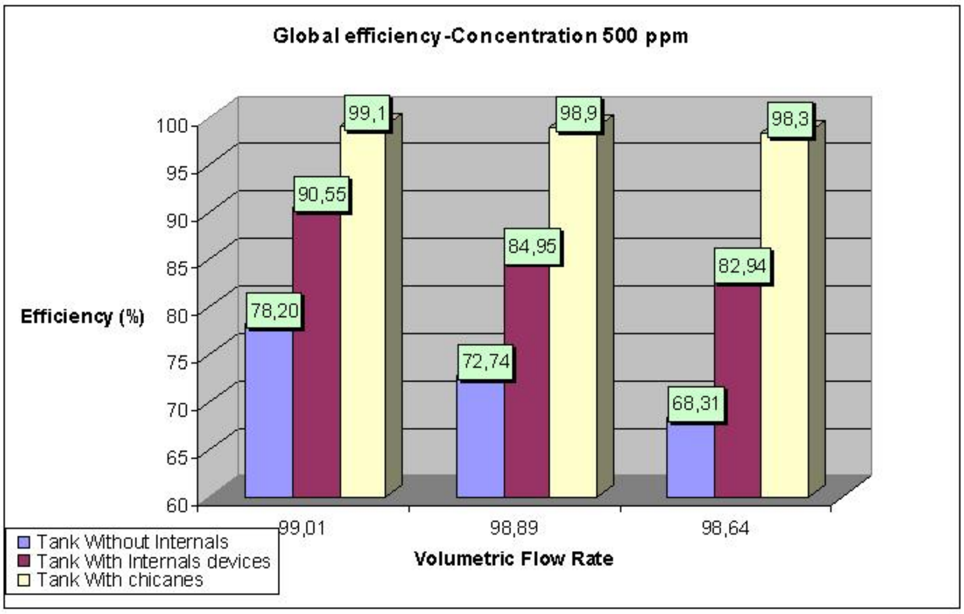

Figure 8. Separation efficiency for $500ppm$ concentration

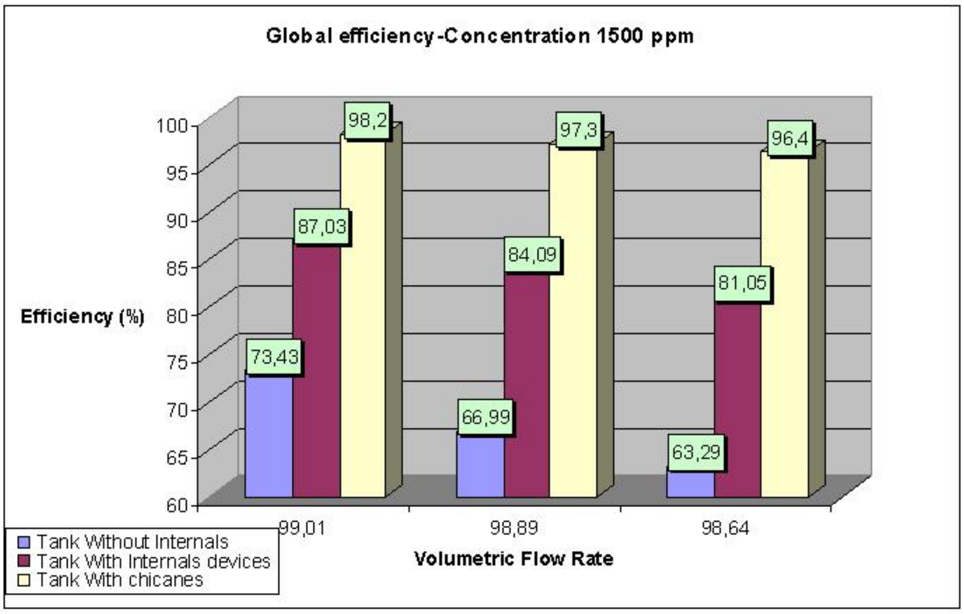

Figure 9. Separation efficiency for $1500ppm$ concentration.

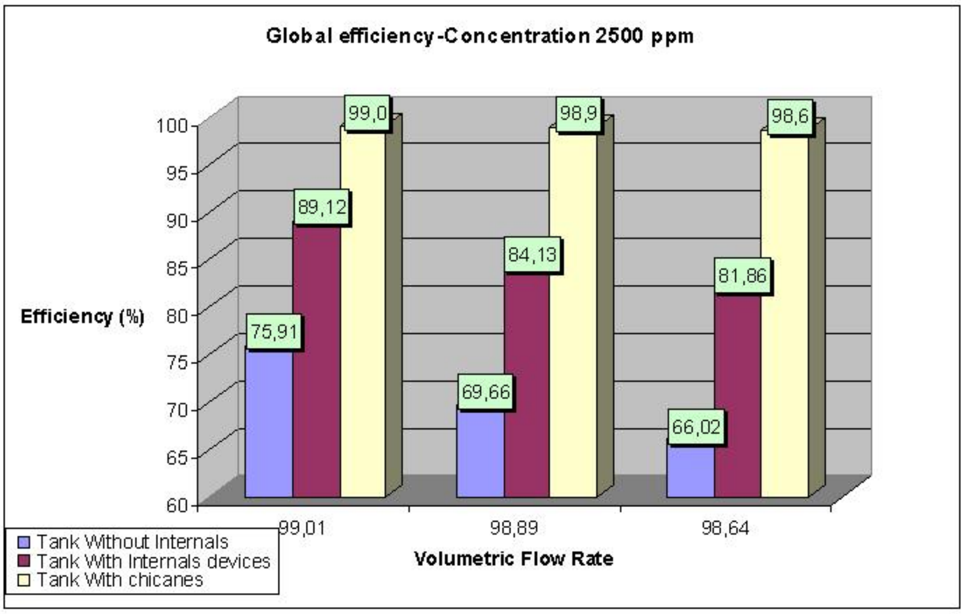

Figure 10. Separation efficiency for $2500ppm$ concentration.

Figure 11, Fig. 12 and Fig. 13 show the efficiencies of separation as function of droplets diameter for the flow rates of 36.000, 67.500 and 90.000 m^3/day. The same trend is observed for all tanks.

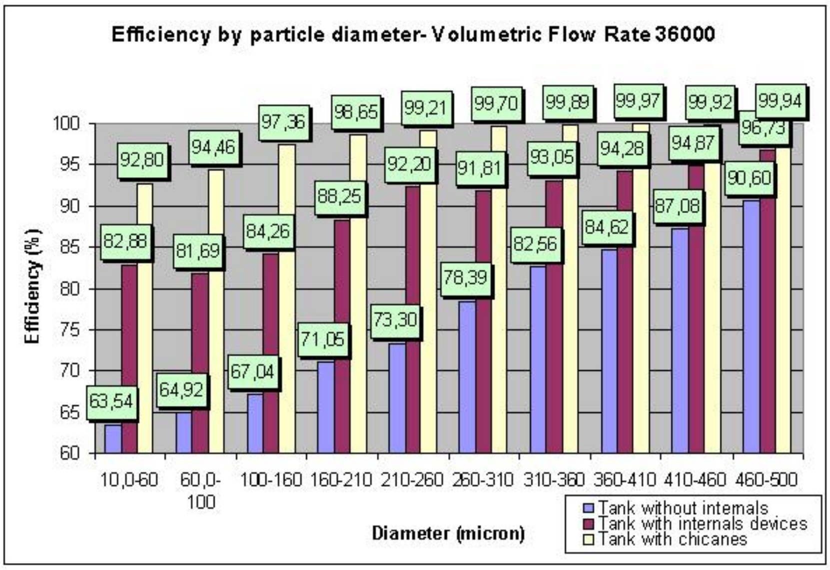

Figure 11. Efficiency by diameter of dropplet for 36.000 m^3/day.

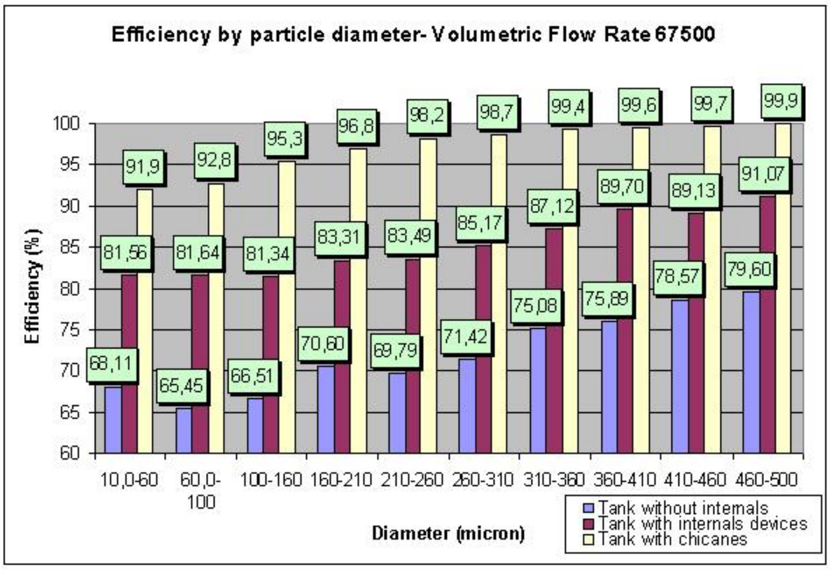

Figure 12. Efficiency by diameter of dropplet for 67.500 m^3/day.

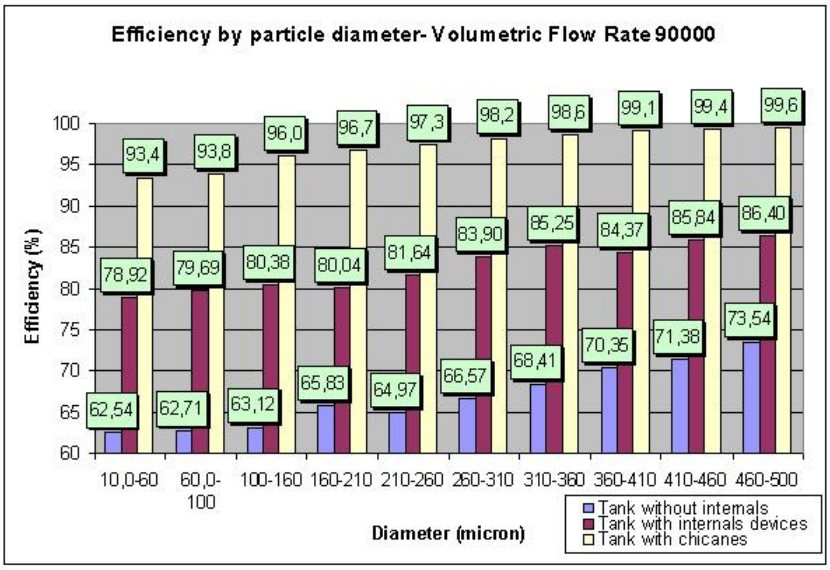

Figure 13. Efficiency by diameter of dropplet for 90.000 m^3/day.

The pictures below justifies why the efficiency of the tanks are so differents in all cases.In the tank without internals, the oily water is released to outlet directly . For other side, the tank with chicanes the oily water is barred by chicanes and all dropplets are collected in two firsts compartments. The tank with internals (collectors and distributor) works well, however, this one has some zones of recirculation that reduces the efficiency.

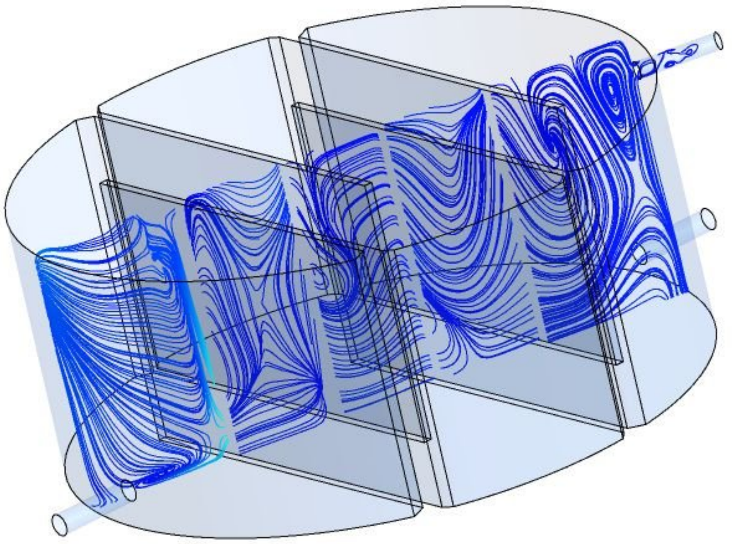

Figure 14. Surface streamlines over the center plane of the tank with chicanes

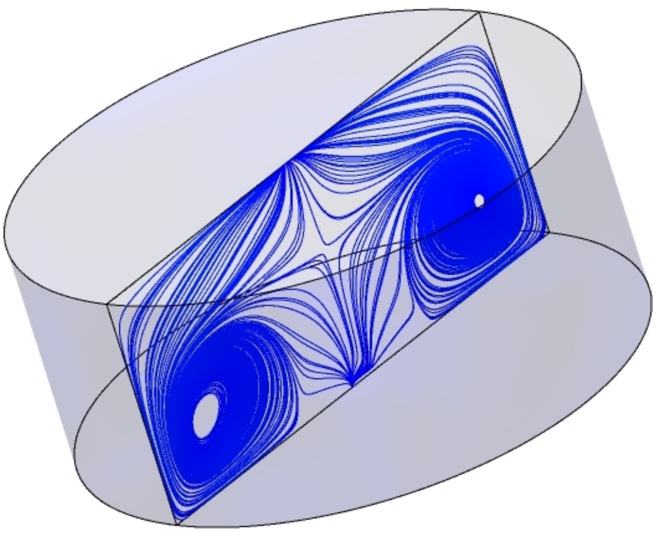

Figure 15. Surface streamlines over the center plane of the tank with internals devices

4. CONCLUSION

For the tank with distributor and collector, the collector of water works inefficiently, because of irregular shape of collection. We can observe that the efficiency of the tank with chicanes was considerably higher compared to the tank with the distributor and collector, while the tank without internals which has the biggest recirculation area and consequently the lowest efficiency.

The presence of baffling is used to reduce all recirculation and dead zones for oil-water separation. We can observe that in pictures 14 and 15. The chicanes breaks the recirculation present in the tank without internals and in the tank with collections and distributor, what explain the high efficiency in the tank with chicanes. Moreover,the methodology was proposed behaved satisfactorily. The computational cost is reduced and the engineers are advised to choose the best configuration that enable to increase the process capacity.

5. REFERENCES

Anastasios I.S,2008, "Improving the hydraulic efficiency of water process tanks using CFD models", Chemical Engineering and Processing: Process Intensification, Volume 47, Issue 8, Pages 1179-1189

ANSYS CFX,2007,“CFX-11 Solver Theory“. Ansys Inc. Canonsburg, USA.

Bird, R.B., Stewart, W.E. and Lightfoot,2004, E.N, "Transport Phenomena. Revised Second Edition".ed ISBN 978-0-470-11539-8.

Crowe, C.,1993,“ Modeling Turbulnce in Multiphase Flows”.

Crowe, C., Sommerfeld, M., Tsuji, Y.,1998,“Multiphase Flows with Droplets and Particle“, CRC Press LCC.

Simmons, M.J.H.,Komonibo, E.,Azzopardi, B.J. ,Dick D.R., 2004 ,"Residence Time Distributions and Flow Behaviour within Primary Crude Oil and Water Separators Treating well head Fluids" Chemical Engineering Research and Design, Volume 82, Issue 10, Pages 1383-1390.