International Journal of Advanced Information Science and Technology (IJAIST) ISSN: 2319:2682
Vol.6, No.9, September 2017 **DOI:10.15693/ijaist/2017.v6i11.69-76**

Exploration of Air Flow inside Oil Tanks by using CFD

K. Vijay Kumar[1], K. Kantha Rao[2],
[1] Asst. Professor, [2] Assoc. Professor,
[1, 2,] Department of Mechanical Engineering,
Samskruti College of Engineering & Technology, Telagana-State, INDIA

Abstract - Computational Fluid Dynamics (CFD) and experimental tests were used to carry out a comparative study of gravity separation using skimmer tank technologies for removing low oil concentrations remaining in produced water. In this paper, three technologies were evaluated; one without internals (called descendent flow technology), second one is with internals (called baffle technology) and the other is with perforate baffles. To determine the flow distribution pattern and residence time. Compare baffle technology model's result with graphical visualization from the literature. CFD results were in good agreement with experimental ones (literatures). However, the suitability of the numerical tools to study residence time was assessed. Velocity magnitude, pressure contours & vectors shows flow characteristics that help to understand and verify the main vortex structures found by CFD leading to the conclusion that the implemented CFD strategies is suitable for evaluation and design of skimmer tanks with long residence time.

KeyWords*:* flow distribution, residence time,

stoke's law, oil tankS, CFD etc…,

1. INTRODUCTION

Produced water quality has become an increasingly large area of concern for the oil production industry. Production facilities have been re-evaluating their conventional approaches to oil removal from water due to increasing water cuts caused by the maturation of their oil wells, as well as a need for cleaner water for re-injection or disposal purposes. As such, the main concerns for producers are that not only do many facilities require an upgrade to their existing equipment to handle higher capacities, but also that their facilities require a more rigorous, reliable system to maintain their water quality for re-injection or disposal specifications. Conventional approaches for water de-oiling include the use of equipment such as gravity skimmer tanks, CPI's, induced gas flotation units, hydro cyclones and filters. Skimmers have been used for a long time, because on their simplicity, low cost and low maintenance.

As oil fields mature, the production of water can significantly increase. The industry perceives this excess produced water as a necessary evil that is often a liability and major cost centre. Offshore platforms are faced with additional challenges. The regulations for oil concentration in produced water discharged overboard commonly vary from 29 to 40 ppm. As the water cut increases, the retention time of existing primary separation equipment is reduced to cope with the excess produced water. Failure to handle the water quickly results in the water treatment becoming the bottleneck of the facility. Reduced retention time in the separation equipment can result in difficulties de-oiling the produced water to within discharge regulations.

1.1 OIL/WATER SEPARATION THEORY

The separation of oil from water and the design of oil/water separation equipment are governed by Stokes' Law which states:

$V_r = g\, d^2 (\rho_w - \rho_o) / 18\, \mu$

V_r = rise velocity of oil droplet
g = acceleration due to gravity
d = oil particle diameter ρ_w =
density of water
ρ_o = density of oil
μ = viscosity of water

From Stokes' Law, several parameters can be manipulated to augment the separation of oil from water. However, the single most effective parameter that helps facilitate the separation process is the diameter of the oil droplet. The manipulation of the oil droplet diameter will have the largest impact on the rise velocity. Other issues play an important role in the separation efficiency process. In order to correctly address the de-oiling issues that exist, it is vital to better understand the characteristics of the produced water and the dispersed oil. Hydrocarbon concentrations should be noted and tracked all equipment that exists for separating oil from water uses Stroke's Law as the basic fundamental operating principle. Each class of oil water separators have their own specific limitation which is the size of the droplet that can be effectively separated from water. The removal of oil from water can be accomplished by the use of several well known and widely accepted techniques. However, the performance of any given separation technique will depend entirely on the condition of the oil-water mixture. Present techniques for the separation of oil from water are based on their difference of density. Stoke's Law states that rising velocity Vr is a function of the square of the oil droplet's diameter.

From Stoke's Law, it can be seen that droplet size has the largest impact on rising velocity to a collection surface and thus the easier it is to treat the water. Consequently, the bigger the droplet size, the less time it takes for the droplet to rise to a collection surface and therefore the easier it is to treat the water.

The oil in water can be present as free-oil, and/or emulsified, and/or dissolved states in different proportions. This oil droplet size distribution is one of the most important factors affecting the design of oil-water separators. Free-oil is defined as an oil droplet of 150 microns or larger which will float immediately to the surface due to its large size and high rise velocity. Emulsion is oil which is dispersed in the water in a stable fashion due to its small diameter and thus to its low rise velocity. Emulsions can be found in two types: mechanical emulsions and chemical emulsions. Mechanical emulsions are created through the process of pumping, large pressure drops through chokes, control valves, and otherwise mixing the oil-water solution. Chemical emulsions are generally intentionally formed using chemicals to stabilize the emulsions for an industrial process need or other use. Gravity separation is a popular mechanism commonly used for the removal of oil from water. This process primarily affects free oil. Tight oil emulsions and dissolved oils will not be removed by gravity separation alone. The objective in treatment of water containing emulsified oils is to destabilize the emulsion so that the oil will separate by gravity or flotation. Essentially what is done is to promote inter droplet contact with the purpose of developing larger droplets that will be easier to remove. Once the emulsion is broken, the same removal techniques applicable to free oil can be utilized. Small oil droplets are always difficult to separate. The smaller the droplets, the lower their rising velocity will be. A prerequisite for efficient separation is, therefore, that oil droplets coalesce (become larger and rise more rapidly). These small droplets and the concentrations of these droplets that exist in the water need to be properly measured in order to select the correct oil/water separator. A Size Distribution Curve is imperative to measure. This information is difficult to obtain if one wishes to measure realistic values indicative of the process. Thus the size distribution curve must be measured in real-time and on-line.

1.2 PROBLEM STATEMENT & OBJECTIVE

Problem Statement

In the present work the numerical investigation and comparative study of gravity separation using skimmer tank technologies for removing low oil concentrations remaining in produced water. In this work, three technologies were evaluated; one without internals (called descendent flow technology), second one is with internals (called baffle technology) and the other is with perforate baffles. To determine the flow distribution pattern and residence time. Compare baffle technology model's result with graphical visualization from the literature.

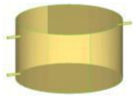
Objective

Computational Fluid Dynamics will be used to carry out a comparative study of gravity separation using skimmer tank technologies for removing low oil concentrations remaining in produced water. Present work aims in three technologies were evaluated; one without internals (called descendent flow technology), second one is with internals (called baffle technology) and the other is with perforate baffles. To determine the flow distribution pattern and residence time. Compare baffle technology model's result with graphical visualization from the literature.

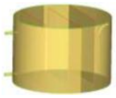
2. DESIGN AND CFD ANALYSIS OF AIR FLOW INSIDE OIL TANKS

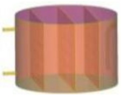
2.1 DESIGN:

Model 1



Model 2 **Model 3**





Dimensions:
Diameter: 1600mm
Height: 1100 mm
Perforate baffles with 24 holes, 30mm Dia. per each plate **the** skimmer model1 and model2 are shared the same geometry and the holes were not included in it. In both cases the holes were represented through mass and momentum sources located at the surface of the vertical duct, model3 included the holes.

Boundary Conditions
Type of flow: Steady-state, K-e model,
Incompressible,
Adiabatic (No Heat Transfer).

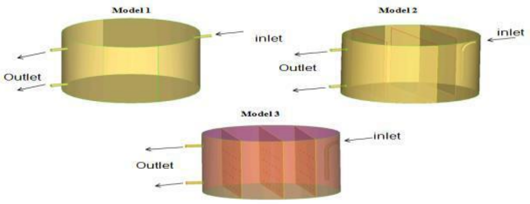


Table-1 Boundary conditions summary:

Condition	Parameter	Model1	Model2	Model3
Inlet	Mass flow rate(l/h)	300	300	300
		500	500	500
Outlet	Pressure (Pa)	0	0	0
		0	0	0

Fluid properties:
Water properties @ ambient conditions

2.2 MESHED MODEL-1

Total number of elements ~ 0.2 million

2.3 MESHED MODEL 2

Total number of elements ~ 0.3million

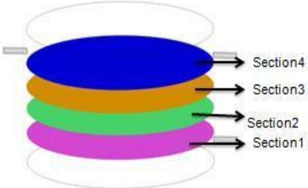
2.4 MESHED MODEL 3

Total number of elements ~ 0.4million

3. RESULTS AND DISCUSSIONS

From this simulation results, the numerical strategy based on flow characteristic was adopted for best capture of results and the significant reduction on time computing. Results also allowing defining the mesh refinement required to capture the more significant flow structures. Flow distribution and pressure calculation were plotted at various plan section below. Section planes height from the bottom of the oil tank.

Section1: 0.240m height from the bottom
Section2: 0.444m height from the bottom
Section3: 0.627m height from the bottom
Section4: 0.820m height from the bottom



3.1. Analysis results for Model-1, 300l/h

Figure 1. shows the velocity field over four horizontal planes placed at the four section plans. For this tank the mesh was significantly refined in order to capture the complex turbulent structures. Results correspond to a mesh with 10e6 grid cells. In pictures velocity magnitude contour was plotted. Stagnation and reciculation flow structures were occuring at the 2and3 section planes.

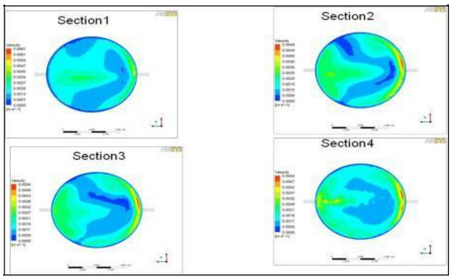


Fig .1 Velocity distribution at various section plans for 300l/h flow rate

Figure 2. display static pressure distribution for different plan sections. At 1and 4 section plane having uniform pressure distribution.Note that statstic presure varires from 0.4878Pa to 0.5008Pa.

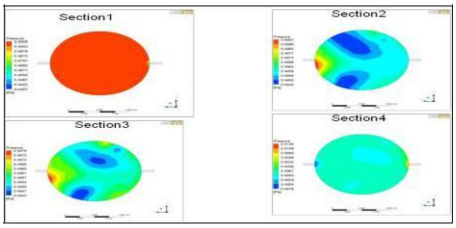


Fig .2 Pressure distribution at various section plans for 300l/h flow rate

Figure 3 shows the velocity vector over four horizontal planes placed at the four plans.Some vortex structures are common along the planes, e.g. the big vortex at the beginning of the second and third section plans.Flow separation at the last vertex internal can be also found in all planes.

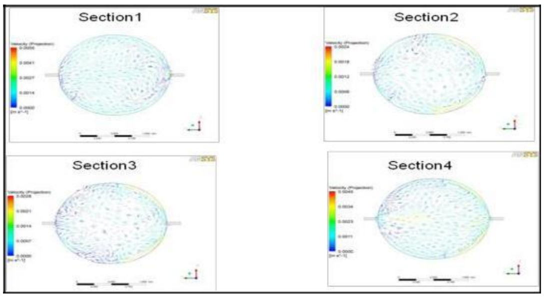


Fig.3 Velocity (vectors) distribution at various section plans for 300l/h flow rate

3.2. Analysis results for Model-1, 500l/h

Figure 4 shows the velocity field over four horizontal planes placed at the four section plansfor 500liter/hours. In pictures velocity magnitude contour was plotted. Stagnation and reciculation flow structures were occuring at all the section planes.

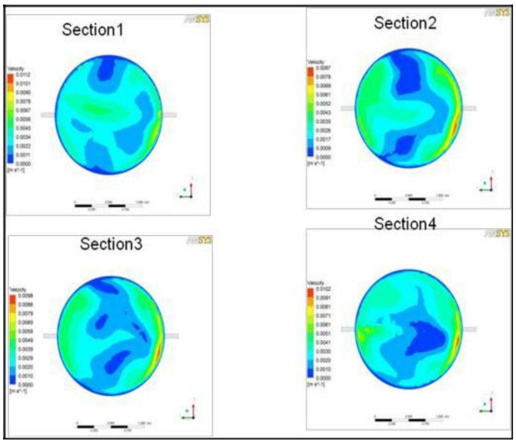


Fig.4 Velocity distribution at various section plans for 500l/h flow rate

Figure 5 display static pressure distribution for different plan sections. At 1and 4 section plane having uniform pressure distribution.Note that statstic presure varires from 1.3370Pa to 1.3726Padue to higher mass flow rate.

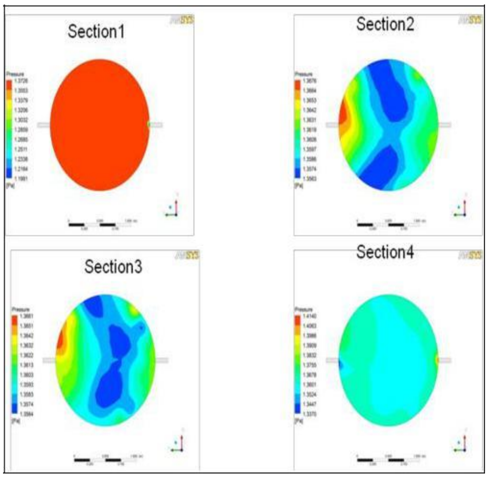


Fig.5 Pressure distribution at various section plans for 500l/h flow rate

3.3. Analysis results for Model-2, 300l/h

Figure 6 shows the velocity field over four horizontal planes placed at the four section plans. For this tank the mesh was significantly refined in order to capture the complex turbulent structures. Results correspond to a mesh with 10e4 grid cells. In pictures velocity magnitude contour was plotted. Stagnation and reciculation flow structures were occuring at all the section planes and also recirculation beginging at comportment2 ,3 and 4.

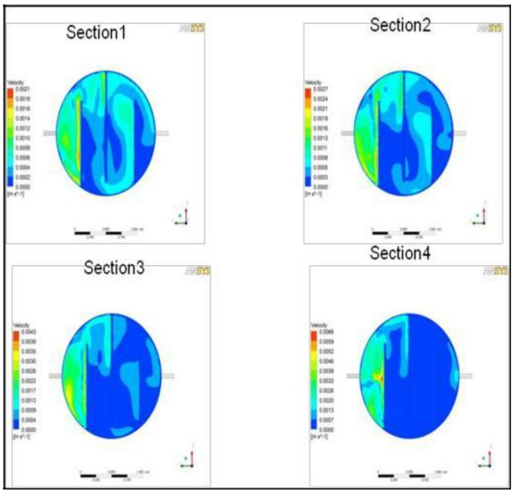


Fig.6 Velocity distribution at various section plans for 300l/h flow rate

Figure7 display static pressure distribution for different plan sections. At 1and 4 section plane having uniform pressure distribution.Note that statstic presure varies from 1.8784Pa to 1.8889Pa. Low pressure obtaining at the first comportments for all the section plans.

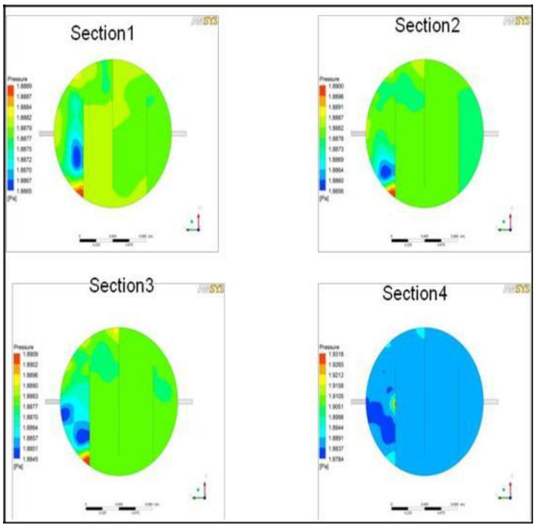


Fig. 7 Pressure distribution at various section plans for 300l/h flow rate

Figure 8 shows the velocity vector over four horizontal planes placed at the four plans.Some vortex structures are common along the planes, e.g. initiate vortex flow at the beginning of the fisrt compartment and grows bigger in rest of the comportments.

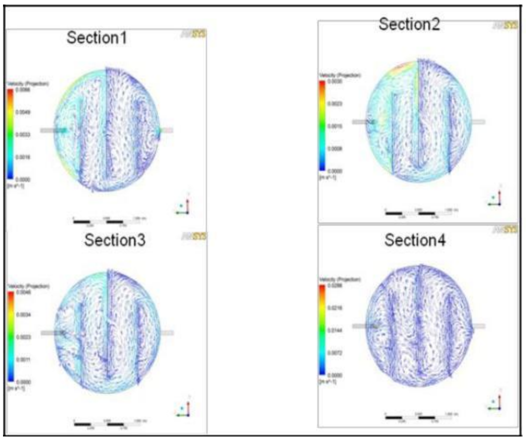


Fig.8 Velocity (vectors) distribution at various section plans for 300l/h flow rate

From Literature(Experimental results), Flow pattern was studied qualitatively by injecting a pulse of a colored tracer (Rhodamine B) and taking pictures at different times. Figure 9, display a sequence of pictures from the upper for different times. Some interesting flow characteristics can be noted. For example, a large recirculation zone is evident where the tracer turn to the third compartment (around the vertex of the central intern).

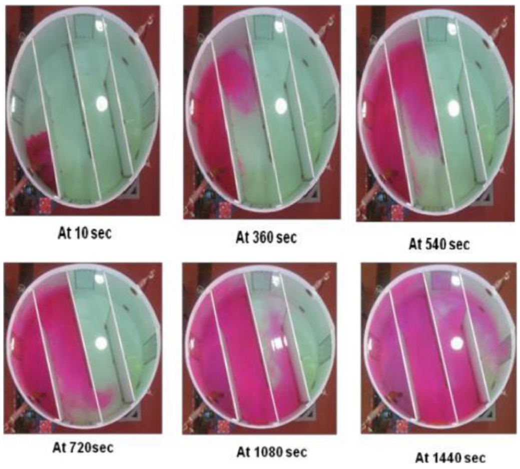
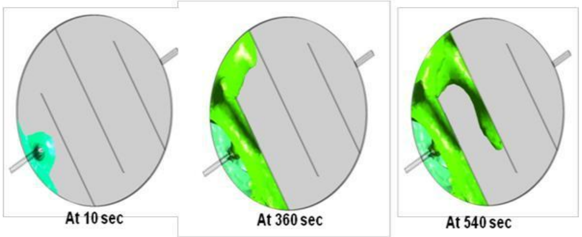

At 10 sec
At 360 sec
At 540 sec
At 720sec
At 1080 sec
At 1440 sec

Fig.9 Residency time ofLiterature (Experimental results), for 300l/h flow rate


At 10 sec
At 360 sec
At 540 sec

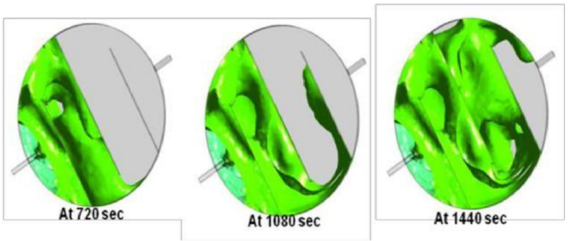

At 720 sec
At 1080 sec
At 1440 sec

Fig.10 Residency time of simulation results, for 300l/h flow rate

Figure10, help to visualize the canalization phenomena from the simulation results. As noted, compared residencial time result with experimental the graghical representation. Age is a measure of local mixing. The residency time is defined as the time which the fluid at a given location has spent inside the oil tanker. Thus, the maximum residency time for the baffle arrangement oil tanker take 1440seconds.

3.4 Analysis results for Model-2, 500l/h

Figure 11 shows the velocity field over four horizontal planes placed at the four section plansfor 500liter/hours. In pictures velocity magnitude contour was plotted. Stagnation and reciculation flow structures were occuring at all the section planes.

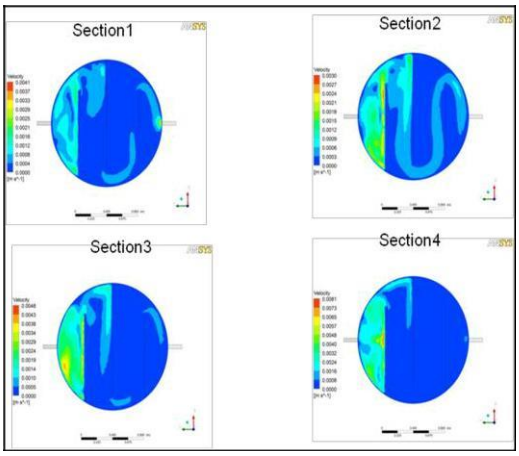


Fig.11 Velocity distribution at various section plans for 500l/h flow rate

Figure 12 display static pressure distribution for different plan sections. At 1and 4 section plane having uniform pressure distribution.Note that statstic presure varires from 0.5857Pa to 0.5943Pa due to higher mass flow rate.

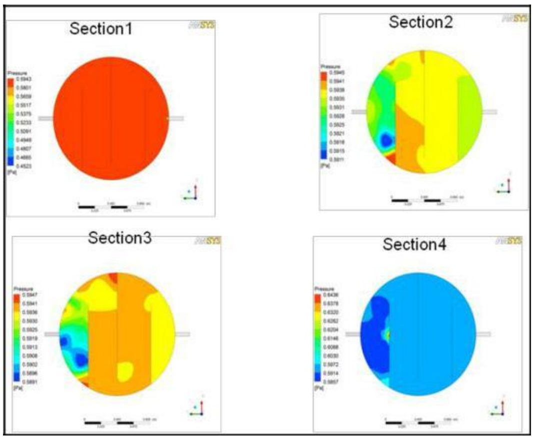


Fig.12 Pressure distribution at various section plans for 500l/h flow rate

3.5 Analysis results for Model-3, 300l/h

Figure 13 shows the velocity field over four horizontal planes placed at the four section plans. For this tank the mesh was significantly refined in order to capture the complex turbulent structures. Results correspond to a mesh with 10e5 grid cells. In pictures velocity magnitude contour was plotted. Stagnation and reciculation flow structures were occuring at all the section planes and also recirculation beginging at comportment2 ,3 and 4.

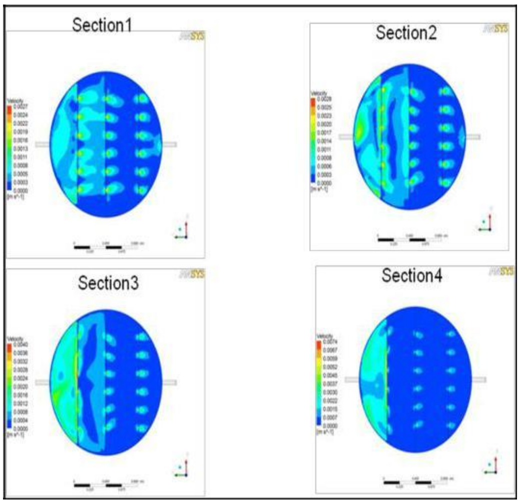


Fig.13 Velocity distribution at various section plans for 300l/h flow rate

Figure 14 display static pressure distribution for different plan sections. At 1and 2 section plane having uniform pressure distribution.Note that statstic presure varires from 0.4995Pa to 0.5274Pa. Low pressure obtaining at the last comportments for all the section plans.

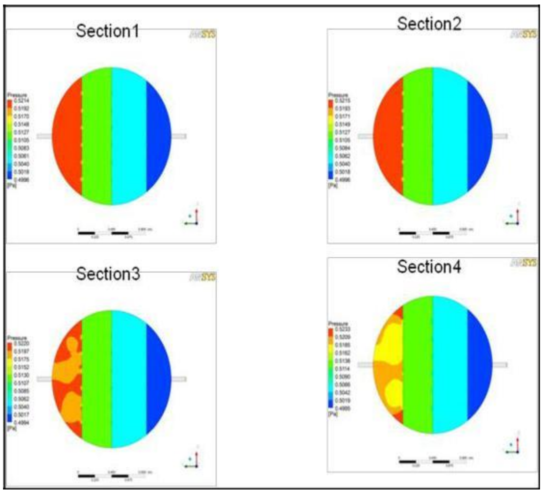


Fig.14 Pressure distribution at various section plans for 300l/h flow rate

Figure15 shows the velocity vector over four horizontal planes placed at the four plans.Some vortex structures are common along the planes, e.g. big vortex flow at the beginning of the third compartment.

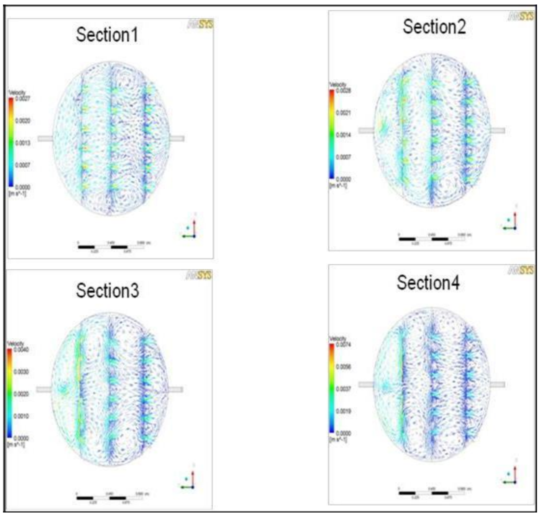


Fig.15 Velocity (vectors) distribution at various section plans for 300l/h flow rate

Figure 16, display a sequence of pictures from the upper for different times. Some interesting flow characteristics can be noted.

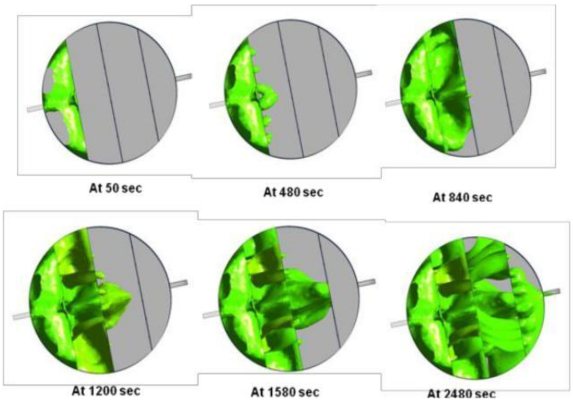


Fig.16 Residency time of simulation results, for 300l/h flow rate

help to visualize the canalization phenomena from the simulation results. This when compared residencial time is almost double with baffle arrangement . Age is a measure of local mixing. The residency time is defined as the time which the fluid at a given location has spent inside the oil tanker. Thus, the maximum residency time for the baffle arrangement oil tanker take 2480seconds.

3.6. Analysis results for Model-3, 500l/h

Figure 17 shows the velocity field over four horizontal planes placed at the four section plansfor 500liter/hours. In pictures velocity magnitude contour was plotted. Stagnation and reciculation flow structures were occuring at all the section planes.

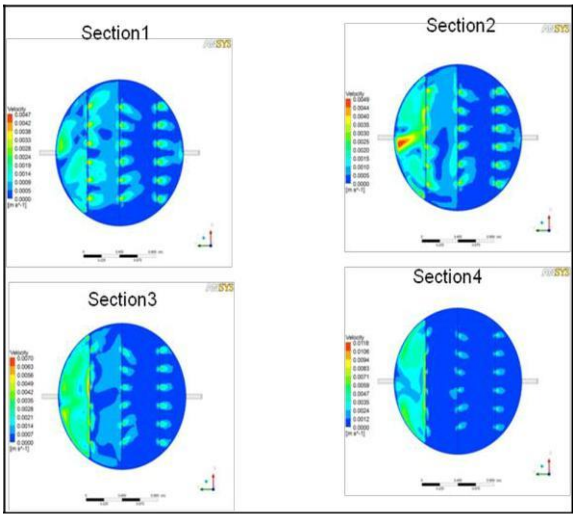


Fig.17 Velocity distribution at various section plans for 500l/h flow rate

Figure18 display static pressure distribution for different plan sections. At section plane-1 having uniform pressure distribution.Note that statstic presure varires from 1.3857Pa to 1.4423Pa due to higher mass flow rate.

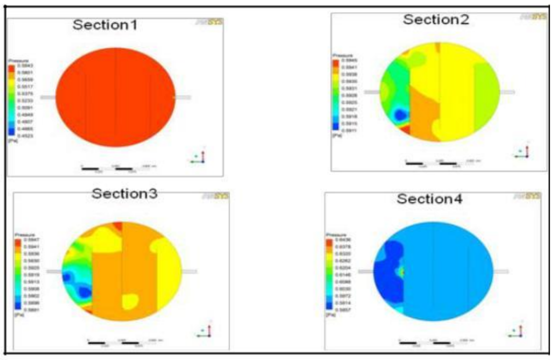


Fig.18 Pressure distribution at various section plans for 500l/h flow rate

4. CONCLUSIONS AND FUTURE WORK

In this work numerical simulation was performed of pilot oil skimmer tank. CFD results were in good agreement with experimental ones (literatures. However, the suitability of the numerical tools to study residence time was assessed. Velocity magnitude, pressure contours & vectors show flow characteristics that help to understand and verify the main vortex structures found by CFD. Based on this simulation he following conclusions was reached:

- Perforate baffles model have more residency time when compared with simple baffle model, it's almost double the time.
- Heavy density particle were settle down on the floor of the oil tank due to long residency time. So that Perforate baffles model is very much suitable for separating the density fluids like oils.

Future work will be devoted to improve the agreement between numerical and experimental results for internal tanks. Additionally, others technologies like vortex flow tanks and coalescence tanks will be studied both experimentally and numerically.

REFERENCES

1) Gimenez J., Ramajo D., Nigro N., Particle Transport in Laminar/Turbulent Flows. MECOM 2012, Salta, Argentina, 2012.
2) Jaworski A.J., Meng G., On-line measurement of separation dynamics in primary gas/oil/water separators: Challenges and technical solutions—A review, Journal of Petroleum Science and Engineering, 68, 47-59, 2005.
3) Jeelani S. A. K. and Hartland S., Effect of Dispersion Properties on the Separation of Batch Liquid-Liquid Dispersions, Ind. Eng. Chem. Res, 37, 547-554, 1998.
4) Kang W, Guo L., Fan H., Meng L., Li Y., 2011, Flocculation, coalescence and migration of

dispersed phase droplets and oil-water separation in heavy emulsion, Journal of Petroleum Science and Engineering, 81, 177-181.

5) Lee C., Frankiewicz T., The Design of Large Diameter Skim Tanks Using Computational Fluid Dynamics (CFD) For Maximum Oil Removal, 15th Annual Produced Water Seminar, Houston, USA, 2005.
6) Lee D. W., Bateman W.J.D, Owens N., 2007, Efficiency of Oil/Water Separation Controlled by Gas Bubble Size and Fluid Dynamics within the Separation Vessel, GLR Solutions, Calgary, Canada, 2007.
7) Wilkinson D., Waldie B., Mohamad M. I., Lee H. Y., 1999, Baffle plate configurations to enhance separation in horizontal primary separators. Short communication. Chemical Engineering Journal, 77, 221-226
8) Zhang L., Xiao H., Zhang H., Xu L., Optimal design of a novel oil-water separator for raw oil produced from ASP flooding, Journal of Petroleum Science and Engineering, 59, 213-218, 2007.
9) ANSYS.ANSYS CFX-Solver Theory Guide, 14th edition, ANSYS Inc., 2015.