Second International Conference on CFD in the Minerals and Process Industries
CSIRO, Melbourne, Australia
6-8 December 1999

ONE EXAMPLE OF HOW OFFSHORE OIL & GAS INDUSTRY TECHNOLOGY CAN BE OF BENEFIT TO HYDROMETALLURGY

David STANBRIDGE[1] and John SULLIVAN[2]

[1] CDS Engineering, Sonsbeekweg 26, 6814 BC Arnhem, THE NETHERLANDS
[2] Sustainable Technologies Australasia, Perth, AUSTRALIA

ABSTRACT

This paper will explore the possibilities of utilising CFD techniques as used in the offshore separation industry to improve the equivalent processes within minerals processing. A test case, specifically in the area of Hydrometallurgy, will be given showing the possible improvements once these techniques are used.

There are many offshore applications requiring very efficient liquid from liquid separation in order to meet strict environmental limitations. These applications involve both the reduction of water in the hydrocarbon phase and the process of reducing the hydrocarbon content within the water prior to the disposal of it to sea. In these disposal applications an entrainment level below 40 ppm is required. By using CFD a vast improvement in separation efficiency can be achieved for which a before and after scenario will be presented and actual field data tabled. It will also be shown that it is not only the separation vessel itself in which the liquid / liquid separation can be improved but also by consideration to the upstream mixing process.

NOMENCLATURE

C'	Drag co-efficient	(dimensionless)
D_d	Droplet diameter	(m)
F_b	Buoyancy force	(N)
F_d	Drag force	(N)
g	Acceleration due to gravity	(m/s^2)
H	Separation height	(m)
L	Separation length	(m)
M_d	Mass of dispersed droplet	(kg)
V_a	Axial velocity of fluid	(m/s)
V_t	Terminal settling velocity	(m/s)
ρ_c	Density of continuous phase	(kg/m^3)
ρ_d	Density of dispersed phase	(kg/m^3)
μ_c	Viscosity of continuous phase	(cP)

INTRODUCTION

CFD techniques have been used widely within the oil and gas market to trouble shoot under performing processes / equipment. One of the specialised areas in which CDS Engineering are involved is that of equipment used for both liquid from gas and liquid from liquid separation.

This paper outlines the modifications to main line separation equipment that led to a substantial reduction in the hydrocarbon within the water phase. A comparison will then be drawn between this process and a Settler tank within a copper production circuit in order to show the savings that could be made if a similar oil & gas market approach was adopted.

BACKGROUND TO OFFSHORE PROBLEM

The offshore application that is to be considered is summarised in the flow schematic below.

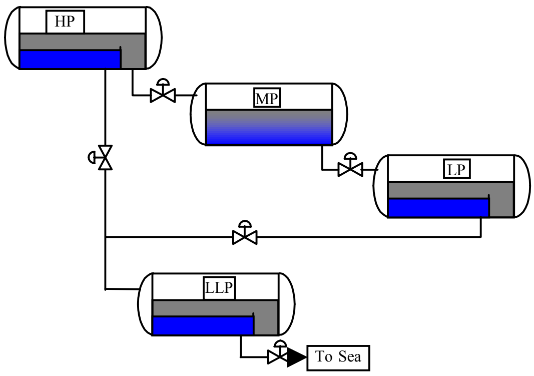


Figure 1: Typical offshore water processing scheme

The system above is typical of an offshore oil production application with 3 stages of pressure let down. Within these systems it is then also general for the HP and LP separators to be 3 phase, that is separating the oil, water and gas. The water streams from these separators can then be treated with further equipment in order to meet the environmental constraints prior to disposal to the sea. Currently this limit in the North Sea is 40 mg/liter.

For the actual system covered by this paper this environmental constraint was not being met without further treatment of the water. It was thus decided to see if something could be done to better the performance. For this system the main loading to the water treatment facility came from the HP Separator with typical figures as shown below:

Date	Entrainment (mg/liter)
18/02/97	473
19/02/97	308
20/02/97	806
21/02/97	619
23/02/97	352
25/02/97	943
31/05/97	835

Table 1: Oil in water entrainment from HP Separator.

In order to find the causes of this poor performance a three-dimensional CFD simulation of the liquid section of the vessel was carried out; the results of which are presented below.

FINDINGS FROM THE CFD SIMULATION

We have given below the summary of the results obtained from the three-dimensional CFD model. In summary, the vessel had a cluster of four inlet cyclones followed by a perforated distribution baffle. However, the perforated plate did not extend fully to the bottom of the vessel and so an even distribution was not found down stream of it.

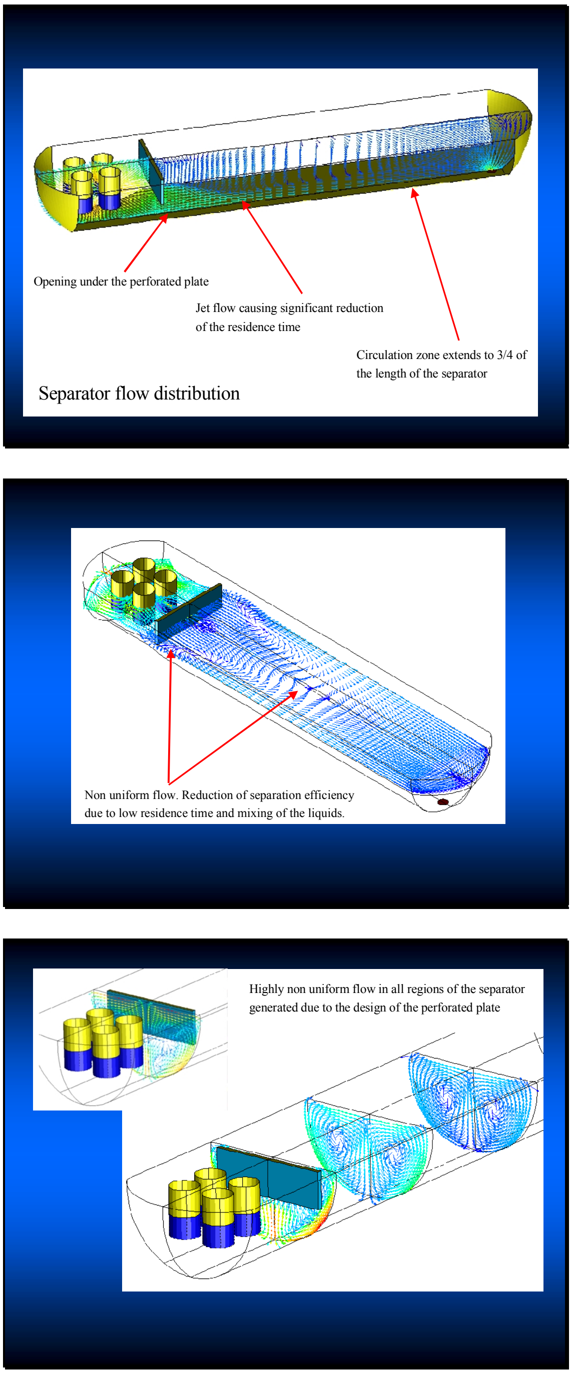


Figure 2: Summary of the CFD simulation of the HP Separator.

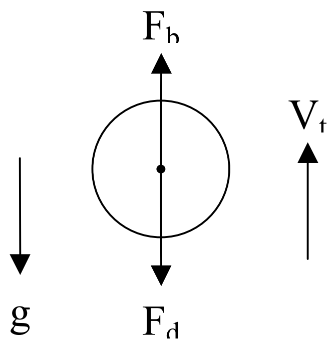
It is easy to see that with the flow distribution shown in the previous pictures optimal conditions for separation have not been set up and therefore better performance should be achievable.

In order to improve the performance of the separator then the jet of liquid flowing inside of the vessel needs to be removed. In this way the theoretical residence time of the fluids will increase thereby maximising the droplet separation within the vessel. Also by removing the jet from the vessel the re-circulation will also disappear that again contributes to an increased residence time of the fluids.

EFFECT OF INCREASED RESIDENCE TIME

By reviewing the equations governing the removal of a dispersed phase from a continuous phase one can see the benefit of increasing the residence time of the fluid.

Ignoring the effects of turbulence and surface tension the forces governing the gravity settling of a droplet are shown in the diagram below. The directions of F_b and F_d relate to whether the droplet is rising or falling. However when determining if a droplet can be separated only the terminal velocity of the droplet is needed. This being the point at which $F_b = F_d$ where steady state conditions exist.



$$F_b = \frac{M_d \left| (\rho_c - \rho_d) \right| g}{\rho_d} \tag{1}$$

$$F_d = \frac{1}{2} C' \rho_c V_t^2 A_d \tag{2}$$

If the droplet is assumed to be a solid sphere then the drag coefficient can be determined from the Reynolds number.

Equating the above gives:

$$V_t = \sqrt{\frac{2gM_d \left| (\rho_c - \rho_d) \right|}{\rho_c \rho_d A_d C'}} \tag{3}$$

Or in terms of droplet diameter:

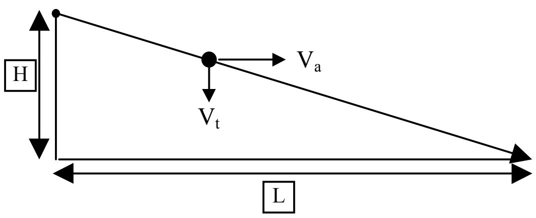
$$V_t = \sqrt{\frac{4gD_d \left| (\rho_c - \rho_d) \right|}{3\rho_c C'}} \tag{4}$$

Rearranging to determine the droplet diameter gives

$$D_d = \frac{3\rho_c C' V_t^2}{4g|(\rho_c - \rho_d)|} \quad (5)$$

For a vessel of a given length the minimum available V_t is determined by the following equation assuming that there is no slip between the continuous and dispersed phase:

$$V_t = \frac{HV_a}{L} \quad (6)$$



The minimum available V_t enables the calculation of the minimum droplet size that can be separated.

Assuming that the Reynolds number of the droplet is less then 1 then Stokes Law gives the following:

$$C' = \frac{24}{\mathrm{Re}} \quad (7)$$

Combining equations (5), (6) and (7) leads to the following relationship:

$$D_d = \sqrt{\frac{0.018\mu_c HV_a}{gL|(\rho_c - \rho_d)|}} \quad (8)$$

Therefore assuming that all parameters in the RH side of equation (8) are constant other than V_a then $D_d = k\sqrt{V_a}$. So if a liquid jet has a velocity 10 times higher than that of the theoretically attainable velocity then the droplet size removed will be approximately 3.2 times as large.

MODIFICATIONS MADE TO THE VESSEL

In order to remove the jet flow from the liquid section the perforated plate was extended to the bottom of the vessel. However due to the location of the liquids entering the vessel, at the bottom of the inlet cyclones, the pressure drop over the perforated plate also had to be increased to ensure that an even distribution was achieved.

With an even flow distribution, the next consideration was to determine the droplet size that can be separated within the vessel. As shown in equation (8) above the main parameters that can be changed to achieve a specified droplet removal are the separation height H, the separation length L and V_a, the axial velocity. However, for this vessel all of these parameters were set by the existing design. Although the performance would have improved just by the amendments to the perforated distribution baffle it was decided to maximise the achievable separation.

Since a better degree of separation was required than could be given by the existing design, further internals were needed within the vessel. These took the form of parallel plates.

As shown in equation (8).

$$D_d = k\sqrt{\frac{H}{L}}$$

So if the ratio H/L is reduced then so is the droplet size removed within the vessel. For the existing vessel H/L was in the order of 0.08. With parallel plates of 10 mm spacing and a length of 2000 mm H/L becomes 0.005 thus providing a 4 times reduction in the droplet size removed. We show in the picture below the arrangement of these parallel plates.

Figure 3: Photograph of parallel plate arrangement.

RESULTS AFTER THE MODIFICATION

After the new equipment had been fitted into the separator the following results were obtained.

Date	Entrainment (mg/liter)
26/08/97	18
27/08/97	23
28/08/97	24
29/08/97	17
30/08/97	6.5
31/08/97	19
08/09/97	22
28/09/97	31

Table 2: Oil in water entrainment from HP Separator after the new internals had been installed.

As can be clearly seen these results show a considerable improvement compared to the original design. This led to the disposal of the water directly to sea, without further treatment.

APPLICATION TO HYDROMETALLURGY

One of the areas that we see can benefit from the approach described above is within the mixer / settler combinations of existing hydrometallurgy plants. In this part of the process substantial savings can be made if the carryover of both the organic and aqueous can be minimised from the settler. Generally, these settler tanks are large with a volume approximately 10 times higher than those installed offshore in order to achieve the same degree of separation. Unfortunately as shown above this large volume does not always guarantee adequate separation.

It should also be noted that within the combined mixer / settler process the mixer tank is providing a sufficiently fine droplet size distribution to ensure that a high degree of mass transfer occurs. The design of the settler therefore must ensure that the droplet size distribution generated within the mixer will be separated within the settler.

In order to see if such improvements are possible within Hydrometallurgy we shall review the design of the new CuSX settler tanks at the WMC Olympic Dam location.

The design comprises a large rectangular vessel with vertical mounted pickets at the inlet end of the tank to act as a distribution device. An outline sketch of this part of the vessel is given below. The sketch is split in two parts, the upper half showing the top view of the vessel whilst the lower gives the relative location of the inlet from the upstream mixer.

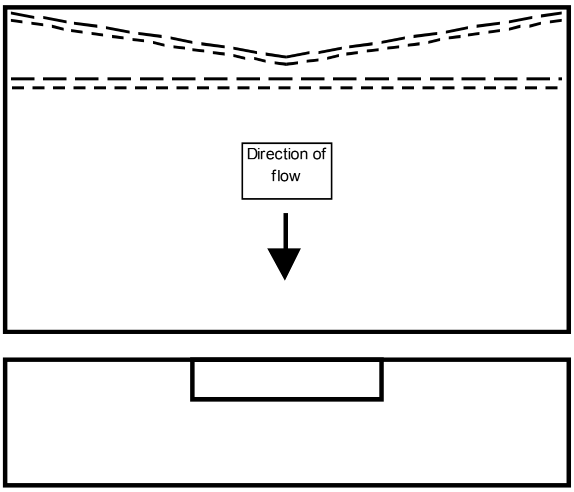


Figure 4: Sketch of the settler inlet configuration

Towards the back of the vessel there is a bucket weir arrangement in order for the organic to leave the vessel followed by an overflow weir for the removal of the aqueous phase.

CFD ANALYSIS OF THE SETTLER VESSEL

As this vessel is essentially a two-phase separator then a two-phase CFD model is preferred allowing for the flow of both the organic and aqueous liquids. However since little is known regarding the flow in these settler vessels a single-phase model will initially be solved in order to see the general flow patterns that are created within the vessel. After this stage then a strategy shall be developed in order to model the vessel with two liquid phases, if necessary.

In order to investigate this vessel in detail a three-dimensional model has been set up using Gambit. The solution itself was obtained using the Fluent 5 CFD code. It should be noted that only one half of the vessel has been modelled since the central vertical axis could be taken as a symmetry plane. The model is shown in the picture below.

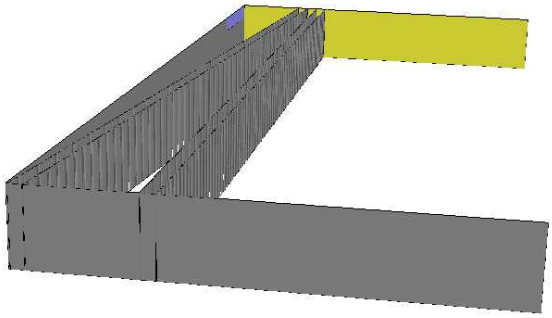


Figure 5: Picture of the settler model generated within Gambit.

The grey areas depict the wall of the vessel, the inlet from the upstream mixer is shown in blue whilst the symmetry is in yellow. What is not clear in the above picture is that each individual picket has been modelled. In this way a very accurate representation of the inlet flow distribution can be achieved within the model, something that is beneficial if any optimisations are to be performed. A close up of the pickets in the CFD model is shown in the picture below.

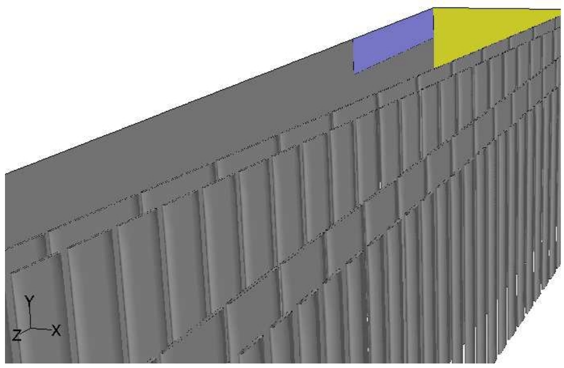


Figure 6: Close up of the pickets clearly showing that each was modelled individually.

FINDINGS FROM THE CFD SIMULATION OF THE SETTLER VESSEL

The result from the simulation shows that the flow in the vessel is not distributed well. What is seen at the inlet is that the flow enters the vessel, strikes the pickets opposite and subsequently starts rotating (see Figure 7). Although the pickets then show a distributing effect the overall effect is not optimal with the simulation showing a velocity range of –0.93 m/s to 1.0 m/s in the flow direction.

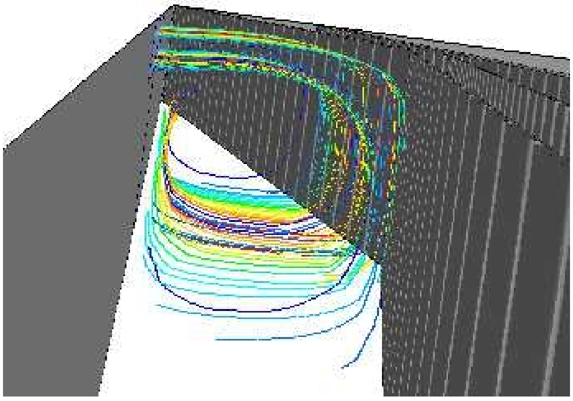


Figure 7: Picture showing the path lines of the fluid after it enters the vessel. A corkscrew motion is clearly seen.

Further figures showing the velocity distribution in the vessel are given below. In all cases the velocity range has been restricted between 0 m/s to 0.06 m/s in the flow direction.

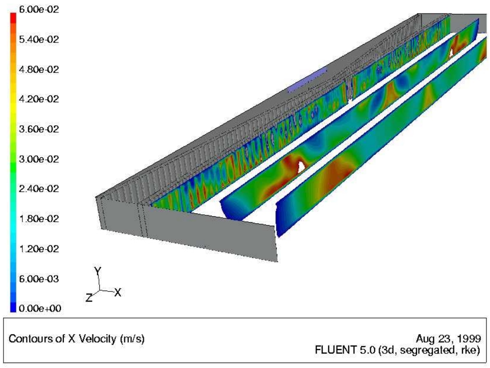


Figure 8: Picture showing the x velocity profile along the length of the vessel.

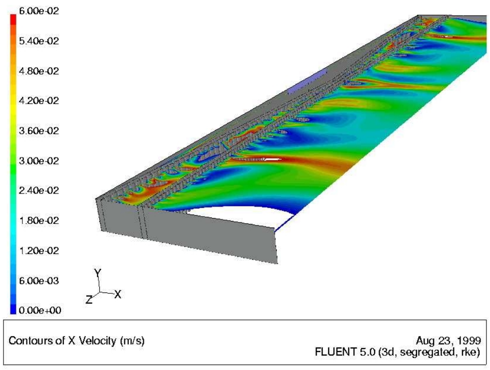


Figure 9: Picture showing the x velocity profile in the plane through the centre of the inlet.

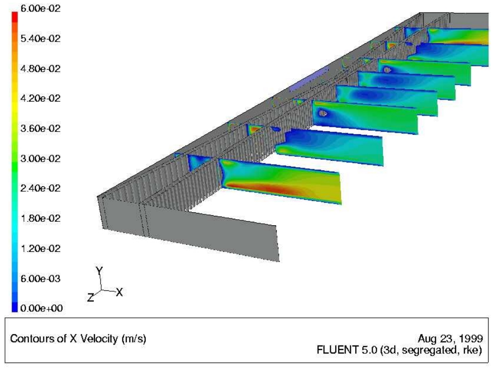


Figure 10: Picture showing the x velocity profile along the length of the vessel.

Although these findings are true only of a single-phase flow we would expect that a similar profile would be found in a two-phase flow simulation of both the organic and aqueous phase. A single-phase simulation was performed initially both due to a quicker convergence and also if the flow distributes evenly for a single-phase it is generally found that it will do so also for a multiphase flow.

In order to prevent the maldistribution shown above then it will be necessary to firstly simulate the settler but with a two-phase flow. After this then a definitive upgrade strategy can be determined in order to solve the maldistribution and hence optimise the gravity separation from the settler.

CONCLUDING REMARKS

From this preliminary investigation it is seen that the flow distribution within the settler is not optimal. This means that increased separation should be achievable if a different arrangement to the currently installed picket arrangement is used.

In order to quantify this further it will be necessary to perform a two-phase simulation of the settler and from there determine a suitable upgrade strategy.