ELSEVIER

Chemical Engineering Journal 77 (2000) 221–226

Chemical Engineering Journal

www.elsevier.com/locate/cej

Short communication

Baffle plate configurations to enhance separation in horizontal primary separators

Derek Wilkinson *, Brian Waldie, M.I. Mohamad Nor, Hsio Yen Lee

Offshore Processing Research Group, Department of Mechanical & Chemical Engineering, Heriot-Watt University, Edinburgh EH14 4AS, UK

Received 23 January 1999; received in revised form 5 October 1999; accepted 21 October 1999

Abstract

Horizontal separators use gravity settling to separate oil from water and other components of oil well riser streams. Inlet designs can induce non-uniform liquid flow across the separator's cross section with a substantial zone of re-circulation. To optimise oil/water separation, the liquid should achieve near to plug flow, so that both phases experience a limited range of residence times. Internal fittings to improve flow uniformity have been investigated.

A single perforated plate across the flow near the inlet increased uniformity of axial velocity across the separator cross section downstream of the baffle. A 10% free area baffle gave better uniformity of flow across the cross section compared to baffles having 5, 15 and 20% free areas. For a fixed free area fraction, the baffle hole size had only a small effect on the flow distribution even though the hole size was varied sufficiently to cover both laminar and turbulent flow through the holes.

Two baffles gave further slight improvement in flow uniformity but only when the baffles were very closely spaced.

Measurements on the flow of a 20% by volume oil in water mixture showed that the flow of this two phase mixture was similarly improved by the presence of a perforated baffle. ©2000 Elsevier Science S.A. All rights reserved.

Keywords: Separation; Primary separator; Oil/water separation; CFD model

1. Introduction

Conventional horizontal cylindrical separators use gravity settling to separate up to four phases (gas, oil, water and entrained solids) from an oil well riser stream. They are generally the largest single item of process equipment in an oil production installation. Reducing their size without loss of separation effectiveness would reduce capital cost and hydrocarbon inventory, thus enhancing both operational safety and economic performance. Separators are large (26 m long by 4.3 m diameter in the case of Chevron's Alba platform in the UK's North Sea oilfield) horizontal axis cylinders which in normal operation are about half filled with liquids. The remaining volume is occupied by hydrocarbon gas.

Small scale experiments and CFD modelling were used to investigate the flow of a single liquid phase in a two-dimensional model separator prior to work on larger three-dimensional models and with two phases, oil and water. Best separation is achieved when the flow of liquids is close to plug flow in the separator without regions of recirculating flow. The aim has been to design simple internal fittings to achieve a plug flow velocity distribution in the separator without inducing further dispersion of the phases.

Having shown reasonable correspondence between CFD and experimental results using a rudimentary two-dimensional model separator without internal fittings, the two-dimensional CFD model was modified to study the effect of various baffles on flow distribution across the cross section. These results have been compared with experimental data obtained in a larger three-dimensional cylindrical model with baffles using flows of water only and of a 20% by volume oil in water mixture. Flow distributions have also been measured with a pair of baffle plates at various spacings in the three-dimensional cylindrical model.

1.1. The flow domains

The two-dimensional rectangular model separator was 0.875 m long by 0.23 m wide and filled to a depth of 0.25 m.

---

* Corresponding author. Tel.: +44-131-449-5111;
fax: +44-131-451-3077.
E-mail address: d.wilkinson@hw.ac.uk (D. Wilkinson).

PII: S1385-8947(99)00170-9

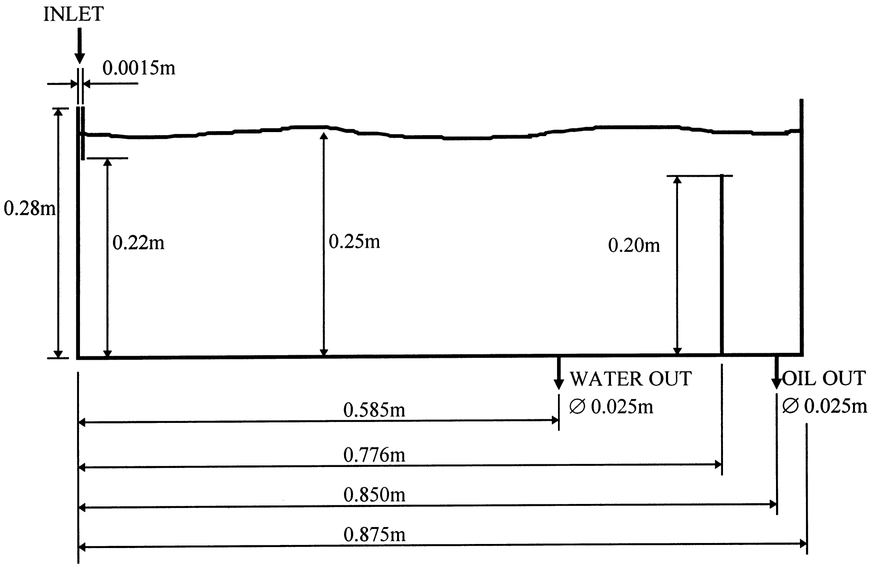

Fig. 1. Side view of two-dimensional separator model.

The inlet was a rectangular slot across the width of the vessel, directed vertically down and emerging below the liquid level in the separator (Fig. 1). This arrangement leads to a significant maldistribution of flow across the separator cross section. Water flowed through the inlet at $0.833\,\mathrm{m\,s^{-1}}$, corresponding to a superficial velocity through the separator of $0.005\,\mathrm{m\,s^{-1}}$. Outlet flow was taken from the 'underflow' or water outlet only. The top boundary was a free surface open to the atmosphere. In the CFD model, the top boundary was represented as a plane wall with zero wall shear stress. In experiments, the top surface was observed to be essentially flat with no significant surface waves, so this was taken to be an appropriate model.

The three-dimensional model was a horizontal axis cylindrical vessel of 0.6 m diameter by 2.26 m long. The fill depth was 0.3 m and the superficial velocity was $0.011\,\mathrm{m\,s^{-1}}$ of either water or an oil and water mixture. The inlet was a circular jet directed horizontally towards a dished inlet diverter above a horizontal flat plate distributor (Fig. 2). The two circular outlets were both fitted with cruciform vortex breakers.

The superficial velocity used in the two-dimensional model gave a Reynolds number of 1575 based on hydraulic mean diameter. The k–ε turbulence model was applied because local velocities were very much higher than the superficial velocity (cf. the inlet velocity and superficial velocity) and the Reynolds number of flow through baffle holes was turbulent in many cases. Reynolds number in the larger cylindrical separator was 6650 based on hydraulic mean diameter and water only flow. The velocities were chosen to give residence times based on superficial velocity similar to practical cases [1] of typically 100–200 s.

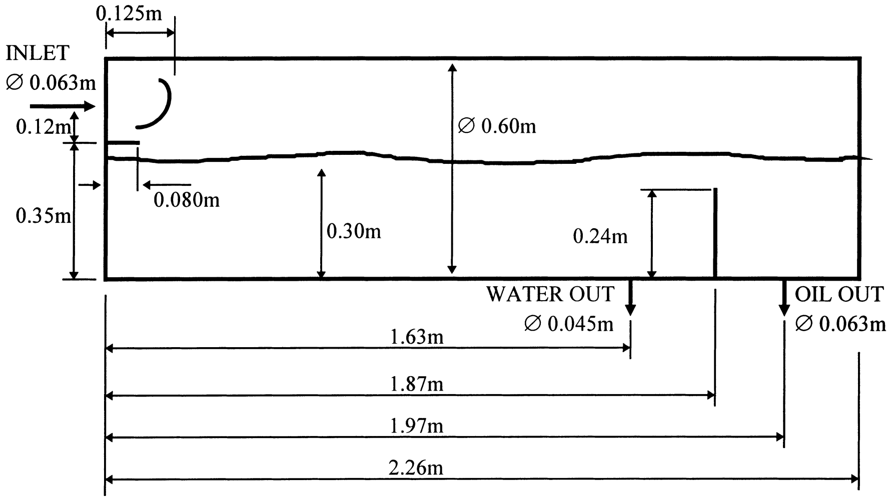

Fig. 2. Side view of three-dimensional cylindrical separator model.

2. Experimental methods

2.1. Two-dimensional rectangular model

The model (Fig. 1) was constructed of acrylic sheet. Detailed fluid velocity measurements were made in the inlet region using Phase Doppler Analyser (PDA) equipment (DANTEC 58N10 PDA processor). Two velocity components were measured at 40 points on a uniformly spaced 25 mm square grid, up to 125 mm horizontally from the inlet and up to 200 mm vertically above the base of the separator. At each grid location, up to 300 separate measurements were made of point velocity. Velocity measurements had a resolution of 1 mm s^{-1} for each component, giving typically 1° and 1.3 mm s^{-1} resolution in the resultant velocity vectors.

A notable feature of the velocity measurements was the large magnitude and long timescale of velocity fluctuations. Typically, the standard deviation of measured velocity at a point was about 40% of the mean velocity. This is reflected in the inconsistencies evident in a few of the measurements (Fig. 3a). In these cases, few of the 300 attempted measurements were successful [2].

2.2. Three-dimensional cylindrical model

A single perforated baffle plate was fitted across the separator (Fig. 2) at 0.3 m from the inlet end. Baffle plates were in 3 mm acrylic sheet. Four of the baffle plates had holes on 40 mm triangular pitch with different hole diameters giving nominally 5, 10, 15 and 20% free areas. Two other baffles having 10% free area were constructed having half the number of holes (22 mm diameter on 56.6 mm triangular pitch) and twice as many holes (11 mm diameter on 28.3 mm triangular pitch) as the standard number of holes. Measurements were also made with no baffle plate fitted. This case was described as 100% free area.

Measurements were made in a plane across the separator at 0.6 m from the inlet using LDA equipment (DANTEC 58N20 FVA). The axial velocity component was measured on a 40 mm square grid of points over one half of a transverse section, 44 points in total.

Measurements were also made with two baffle plates fitted, both having 10% free area but with differing hole pitches, so that holes in the two baffles were not in alignment. One baffle was positioned at 0.6 m from the inlet end and the location of the other was set successively at several positions from 0.1–0.55 m from the inlet end. With two baffle plates, measurements were made using Acoustic Doppler Velocimetry equipment (Nortek ADVField) positioned 0.3 m downstream of the downstream baffle plate. Again, the axial velocity component was measured on a 40 mm square grid of points over one half of the cross section.

Velocities were observed to be subject to large scale fluctuations so some measurements were taken over a range of sampling times to establish the minimum time necessary to obtain a representative time averaged velocity at each point. The graph (Fig. 4) shows clearly that a sampling time longer than 100 s was required to give a representative measurement. At each location, therefore, measurements were made over 120 s, giving typically 1000 individual measurements. Velocity resolution was ±0.03 mm s^{-1}.

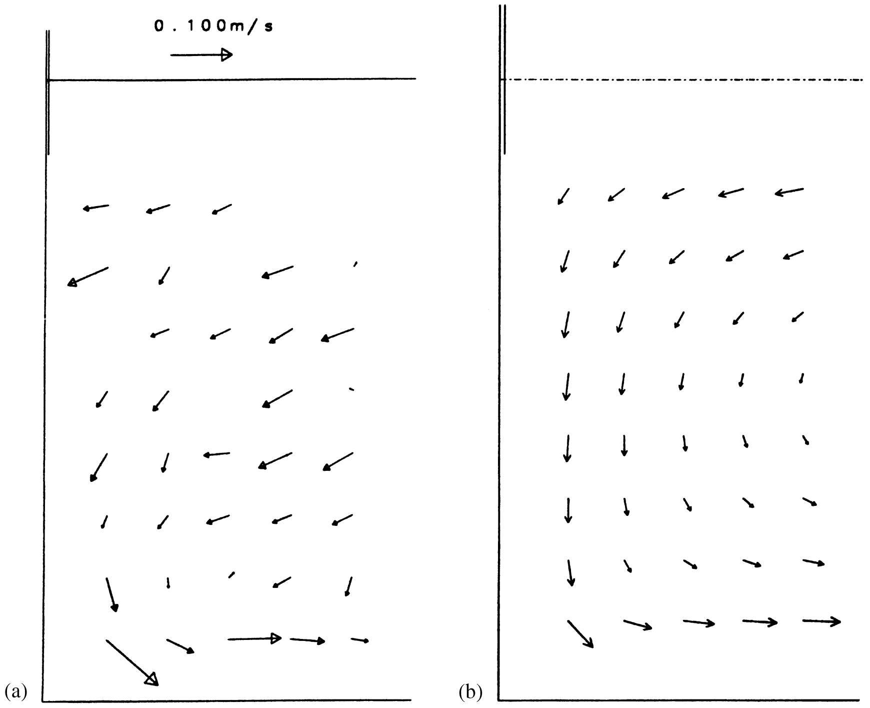

Fig. 3. Velocity vectors in the two-dimensional models: (a) measured; (b) CFD results.

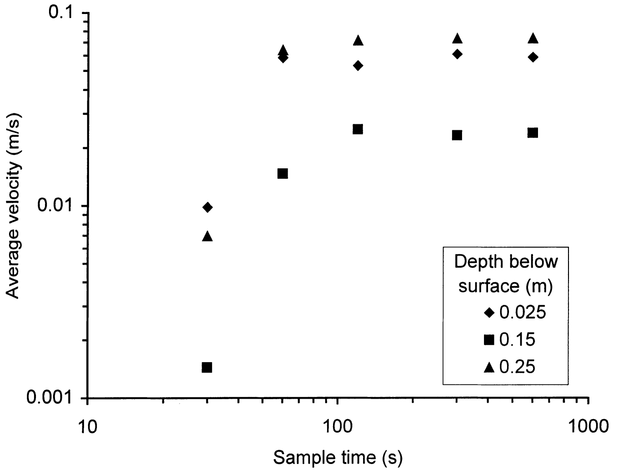

Fig. 4. Effect of sampling time on average velocity measured 100 mm downstream of baffle.

2.3. The CFD model

Flow in the two-dimensional model was simulated using the k–ε turbulence model even though the flow Reynolds number is quite low based on hydraulic mean diameter and superficial fluid velocity of the system. This is because local velocities were very much higher than the superficial velocity. The flow was defined to be two-dimensional, steady, isothermal and incompressible.

The baffle, weir and inlet side plate were described as vanishingly thin plates in the CFD model, although these were made of 3 mm thick acrylic sheet in the physical models. The two outlets were circular holes of 0.025 m diameter fitted with conventional cruciform vortex breakers in the physical model. The underflow outlet was depicted in the CFD model as a slot of width 2.1 mm to give the same mean velocity through the outlet in the CFD model as existed in the experiments. The three-dimensional separator has not been modelled by CFD in this study. An aim has been to investigate the value of a simple two-dimensional simulation of the three-dimensional separator.

The CFD model without baffles was constructed using the Menu system of PHOENICS V1.5 [3]. The resulting PIL (PHOENICS Input Language) file was then edited to incorporate baffles. The problem was solved on a 90 MHz Pentium PC.

The inlet was specified as a plug flow of water at 0.833 m s^{-1} directed vertically downwards. The inlet turbulence intensity was set at 0.02%. The underflow outlet was specified as a constant pressure outlet. The CFD model's outlet was located two cells, 0.01 m, below the plane of the separator's base. Walls were specified at both ends. The base of the model was a fully blocked (zero porosity) region with wall friction. The two internal plates at inlet and weir were specified as zero porosity cell faces with wall friction. The free surface was not specified explicitly since PHOENICS defaults to an impermeable frictionless boundary. Baffles were simulated using discrete lengths of zero porosity cell faces with wall friction. Holes in baffles were either one or two grid cells wide. Further details of the CFD model were given previously [4].

Validity of the CFD solutions has been substantiated by repeating solutions for several configurations using significantly more grid cells, with different levels of inlet turbulence and using a more rigorous convergence criterion. It was found that the effects of these changes on the results were not significant [4].

3. Results and discussion

The results obtained by CFD for the plain two-dimensional vessel without baffles are summarised in the vector plot, Fig. 3b. Both the geometric and vector scales are the same as in Fig. 3a to facilitate comparison.

The effectiveness of baffles in improving uniformity of the flow has been assessed by comparing the standard deviation of time averaged axial velocity measurements at points across the cross section. In the CFD model, this was evaluated at the plane 0.5 m downstream of the inlet. For the three-dimensional model, the standard deviation was calculated of axial velocities measured across the cross section 0.3 m downstream of the last baffle. In all cases, the standard deviation calculations apply to measurements at a distance equal to the fill depth downstream of the last baffle.

The solid line in Fig. 5 shows the effect of varying baffle free area on the standard deviation of velocity in the two-dimensional CFD model. Experimental measurements in the cylindrical vessel with one baffle are shown by triangular symbols. The hole diameter for each case is shown alongside the symbols.

It can be seen from the three results obtained using a baffle with 10% free area that there is an effect of hole size on baffle performance but this is much less marked than the effect of baffle free area. Data gathered at other locations in the separator indicate a similar effect of hole size on the standard deviation of velocity to that which is shown by Fig. 5.

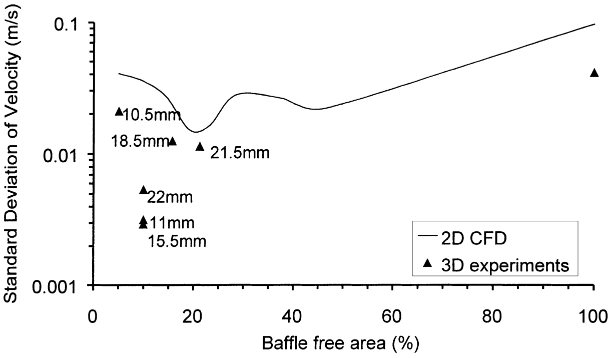

Fig. 5. Standard deviation of velocity across the separator with one baffle-experimental and CFD results.

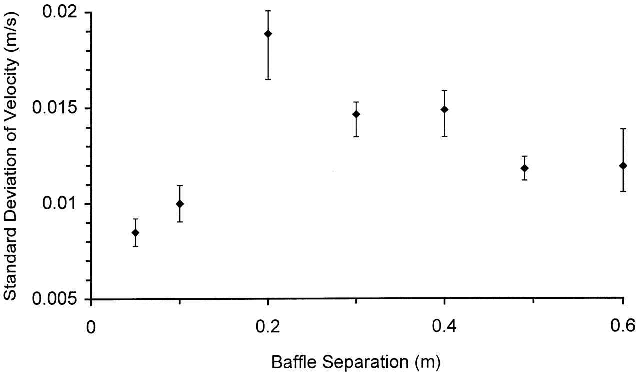

Fig. 6. Standard deviation of velocity across the separator with two 10% baffles-effect of baffle spacing.

Experiments performed in the cylindrical vessel using two baffles both having 10% free area showed no significant extra effect due to the second baffle except when the separation between the baffles was less than 0.1 m (Fig. 6).

All the experiments reported so far were conducted using water only. Two runs were performed to investigate the effect on flow patterns of an oil/water mixture. In these, the inlet fluid was a mixture of 20% by volume of a mineral oil blend (density 805 kg m^{-3}, viscosity 0.014 Pa s) with water. The oil phase was in the form of drops having mean diameter 40 μm. This was measured, after dilution, by laser diffraction (Malvern Mastersizer). Two cases were studied, no baffle and a 10% free area baffle (11 mm holes). From a comparison of the two cases, Table 1, it was concluded that measurements in water only flows give a useful indication of the flow in separators containing both oil and water phases. This result concurs with the findings of Hwang and Pal [5] that pressure loss coefficients for the flow of oil/water mixtures through sudden expansions and contractions are not significantly influenced by the concentration or type (oil in water or water in oil) of the mixtures.

3.1. Comparison between CFD and experiments

The pattern of flow in the inlet region of the unbaffled two-dimensional CFD model is a good description of the experimental measurements (Fig. 3).

At first sight, comparing the effect of different baffles in the CFD model with experimental results in three-dimensions is discouraging (Fig. 5). The experimental finding that a 10% free area baffle gives lower standard deviation of time averaged axial velocity (a better separator) than the 5, 15 and 20% baffles has a counterpart in the minimum of standard deviation at 20% free area displayed by the two-dimensional CFD model. However, the optimum suggested by CFD is at a different value of baffle free area. There is also a significant difference in the magnitudes of standard deviation between two-dimensional CFD and three-dimensional experimental results. But the CFD model does identify that there is an optimum value for baffle free area to maximise flow uniformity. The cases under consideration are significantly different, the CFD model being smaller and two-dimensional, so some discrepancy is to be expected.

Table 1
Standard deviation of velocity at 0.6 m from inlet (m s^{-1})

Fluid	Water	20% oil in water
No baffle	0.0267	0.0411
10% free area baffle	0.0031	0.0028

3.2. Previous work

Taylor and Batchelor [6] reported that, for steady flow of air through a wire gauze, maximum suppression of non-uniformity in longitudinal velocity is achieved with a free area of 45–50%, independent of wire diameter. In suppressing turbulent fluctuations, however, it was reported [6] that there is no optimum free area but there is a monotonic reduction in the level of fluctuations as free area is reduced.

Perforated plate flow conditioners designed to improve flow distribution upstream of an orifice plate flow meter typically have 50–60% free area but these generally employ a non-uniform distribution of holes of several sizes to encourage the formation of a fully developed pipe flow velocity distribution [7].

Hansen et al. [8] described a collection of computer codes for flow modelling in primary separators and an associated experimental study. Their model incorporated a perforated baffle plate. Results were given for one pattern of baffle plate having 4% free area. Experiments were performed in a 1.83 m long by 0.46 m wide rectangular vessel. Axial velocity measurements at the centreline were compared with computer predictions. They also used residence time distribution to indicate the degree of uniformity of the velocity distribution.

3.3. Drop breakage by baffles

A potential drawback associated with perforated baffles in separators is the breakage of drops of oil (or drops of water at high oil concentration) by turbulence and other effects generated by increased velocity through the perforations [9]. Since smaller drops are more difficult to separate, the benefits of improved flow distribution could be negated. Drop breakage during flow through perforations has been measured and the results were correlated in terms of energy dissipation based on Kolmogorov's model of turbulence [9]. At the perforation velocities studied here, drops in the size range relevant to primary separators would, according to that correlation, not suffer breakage. In a full scale separator where velocities are higher, particularly a floating production system in which marine motion can increase local internal velocities by up to 10 times, breakage is more likely. A further optimisation of baffle free area would then be required.

4. Conclusions

The CFD model provides a reasonably good simulation of the flow measured in the two-dimensional experimental model. There is also an interesting parallel between the degree of flow smoothing and baffle free area found in the two-dimensional CFD model and that measured in the three-dimensional experiments. While the CFD model does not predict the optimum baffle free area accurately, it does indicate the existence of an optimum value to maximise flow uniformity. Thus, the simple two-dimensional CFD model is felt to be of some use in studying ways to improve the performance of baffled separators.

The effect of varying the size of baffle holes while maintaining the baffle free area has been found to be comparatively slight. A restricted investigation of the flow of oil/water mixtures suggests that results obtained for a single phase flow can give a useful guide to behaviour to be expected in the two phase flow case.

Two baffles are in general no more beneficial than a single baffle unless they are spaced closer than 0.1 m apart.

Several parameters of the problem remain to be explored. In particular, the effect of adding further baffles remains to be investigated. It is also proposed to simulate by CFD the large scale three-dimensional separator. Further work will also involve two phase studies both by experiment and CFD. It has yet to be verified that the more common separator operation with a continuous oil phase is similar to operation with a continuous water phase.

Acknowledgements

The authors are grateful to Conoco UK, who supplied funding for the LDA apparatus used here through a previous project.

References

[1] K. Arnold, M. Stewart, Surface Production Operations, Vol. 1, Gulf, Texas, 1991.
[2] D. Wilkinson, B. Waldie, Trans IChemE 72, Part A (1994) 189–196.
[3] D.B. Spalding, J. Ludwig, H.Q. Qin, CHAM Report TR200. PHOENICS Reference Manual CHAM, London, 1989.
[4] D. Wilkinson, B. Waldie, M.I. Mohamad Nor, Improving oil/water separation in primary separators, World Conference on Particle Technology 3, Brighton, 1998.
[5] C.-Y.J. Hwang, R. Pal, Chem. Eng. J. 68 (1997) 157–163.
[6] G.I. Taylor, G.K. Batchelor, Quart. J. Mech. Appl. Math. 2, Part 1 (1949) 1–29
[7] E.P. Spearman, J.A. Sattary, M.J. Reader-Harris, Flow Meas. Instrum. 7 (1996) 181–199.
[8] E.W.M. Hansen, H. Heitmann, B. Lakså, A Ellingsen, O. Østby, T.B. Morrow, F.T. Dodge, in: A.P. Burns (Ed.), Fluid Flow Modelling of Gravity Separators in Multiphase Production, Elsevier, London, 1991.
[9] B. Waldie, 1996. Separators for floating production, Proceeding of Conference on Production Separator Developments, Aberdeen, 1996.