Separation Offshore Survey – Design/redesign of Gravity Separators

a report by

Ernst W M Hansen

General Manager, Complex Flow Design AS

Separation Offshore

The separation process in petroleum production involves a number of equipment units. In many fields the process is composed of a number of separation steps where the pressure is stepwise reduced and gas flashed off. All oil, gas, water and sand separation is based on differences in density and most separators are gravity separators, i.e., they utilise the common acceleration of gravity, in huge pressurised vessels. Later developments have brought compact, reliable high-*g* equipment to the market, particularly hydrocyclones and centrifuges for oil and water polishing. The fluid flow behaviour of a three-phase separator shows that different physical and chemical phenomena are important in different zones.

Separator sizing must satisfy several criteria for good operation during the lifetime of the producing field. It must:

- provide sufficient time to allow the immiscible gas, oil and water phases to separate by gravity;
- provide sufficient time to allow for the coalescence and breaking of emulsion droplets at the oil–water interface;
- provide sufficient volume in the gas space to accommodate rises in the liquid level that result from surge in the liquid flow rate;
- provide for the removal of solids that settle to the bottom of the separator; and
- allow for variation in the flow rates of gas, oil and water into the separator without adversely affecting separation efficiency.

Horizontal Gravity Separators

Each separator operates at a fixed pressure. The multiphase fluid enters the vessel through the inlet nozzle as a high momentum jet hitting an inlet arrangement such as a momentum breaker device or cyclones. The liquid is diverted into the liquid pool in the lower part of the vessel. The low-density gas, together with liquid drops, will flow in the upper part of the vessel. Inside the liquid pool, near the inlet, the multiphase fluid flows as a dispersion with low horizontal velocity. The low-density gas and mist rises and the oil, water and emulsion will separate by gravity on the way to the outlet. In the upper part of the vessel, the oil and water particles together with condensed gas will fall down to the liquid interface as the multiphase fluid flows to the outlet. For the modelling and simulation of fluid flow and phase separation behaviour inside a gravity separator, it is helpful to characterise the different flow regimes or zones (see *Figure 1*).

Fluid Systems and Mechanisms of Separation

The separation process is dominated by two factors:

- the emulsification process, which takes place in the choke and other equipment components with high shear; and
- the coalescence and settling effects, where drops grow and settle or cream to its homo-phase.

The Bottle Test (Batch Separation)

A common method of determining relative emulsion stability of water-in-oil or oil-in-water is a simple bottle test (batch separation test). These batch sedimentation and coalescence experiments are an attractive method for the study of separation behaviour of emulsions due to its simple and inexpensive experiments. When the percentage of water in the mixture exceeds approximately 20% by volume, two interfaces become visible as the mixture settles. One is a sedimentation interface between the settling dispersion and the bulk oil phase. The other is a coalescence interface between the dispersion and the bulk water phase. As time goes on, the thickness of the dispersion layer grows smaller and the two interfaces approach each other.

The mixture in a batch settler behaves as a quasi-homogeneous flow and may be described by an

Figure 1: Typical Flow-zones in a Horizontal Gravity Separator

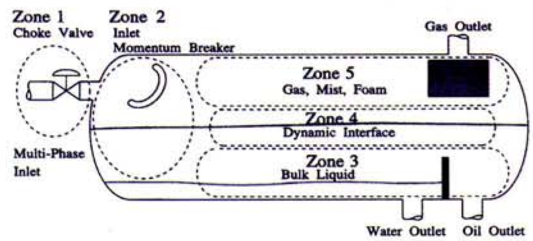


Figure 2: Drop-size Distribution for an Oil/water Mixture

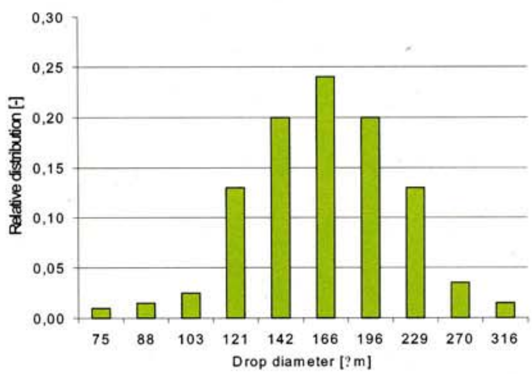
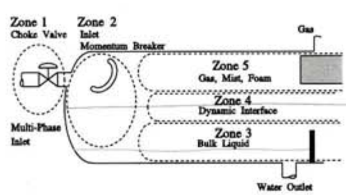


Figure 3: Simulated Results of Separation Interfaces in Batch Settling



advanced mixture model, such as the drift–flux model. The drift–flux model is one of the multi-phase flow models in the commercial CFD programme FLOW-3D.

The driving forces in the collision frequency are the settling of drops, the shear flow and the turbulence in the flow-field.

Modelling and Simulation of Batch Separation

The mathematical modelling of coalescence in unstable oil-in-water emulsions is thoroughly described in the literature. Numerical simulations for batch settling of a poly-dispersed water-in-oil emulsion are performed and the results are presented in *Figure 3*. The water-cut in the emulsion was 20%, the density difference of the two fluids was 200kg/m^3 and the height of the settling container was 0.3m. The drop-size distribution (see *Figure 2*) was initially configured in 10 different drop-classes. The Sautern mean diameter in the distribution is 160μm. Settling in the batch was simulated with and without a coalescence model and, as seen in *Figure 3*, the separation ran faster with the drop-growth model (coalescence model) included. The separation time for the current batch system is 430 seconds, calculated by Stoke's unhindered settling equation.

The separation time is 1,300 seconds calculated by the hindered settling equation from Kumar and Hartland, 1985. The simulated result with a constant Sautern mean drop-size of 160μm (original drift–flux model) gives a long settling time of approximately 1,800 seconds. The simulated result with a drop-growth model shows the approach of the two interfaces after 1,150 seconds, which is in relatively good agreement with the hindered settling equation.

Hydraulic Behaviour in a Horizontal Separator

The gas/liquid and liquid/liquid systems will result in two relatively separated flow systems in pipelines or separators, due to their density differences. The gas and liquid flows with rather high velocities enter through pipe and inlet device to the separator. Inlet devices, such as cup-shaped plates, turbine–vane arrangements or cyclone arrangements are normally mounted as momentum breaker devices. The momentum breaker device will lower flow velocities and separate the gas and the liquid phases quickly with a minimum space required in the longitudinal direction. The flow leaving the momentum breaker will further introduce the gas (gas and mist) and liquid phases (water and oil) into gas volume or liquid pool of the separator, respectively. A horizontal three-phase separator is normally about half-filled with liquids (oil/water) to allow the gas to evacuate with a proper retention time in the upper part of the separator.

Fluid Flow Modelling and Simulation in a Test Separator

The test separator is a conventional horizontal gravity separator with a diffuser and a cascade tray as the inlet arrangement (see *Figure 4*). The inlet arrangement is mounted on the inlet riser, which is located centrally at the end. The separator is a three-phase separator, (i.e., separation of gas, oil and water). The separator is about half filled with liquids during operation, overall length is 11.5m and the diameter is 3m. The flow model covers the flow from inlet to the weir plate in the separator. The fluid flows leaving the momentum breaker device have to be estimated and introduced as an inlet flow condition to the bulk flow zones in the horizontal separator.

The normal liquid level is 1,265mm and the normal interface level is at 942mm. The oil zone thickness is 323mm – thus most of the liquid volume during operation is filled with water. The water cut in this study is 29.3%. The flow pattern in vertical slices, from the inlet to the weir, is presented in *Figure 5.* The flow upstream of the distributor plate is governed by the flow from the inlet. The flow from the inlet and the restriction of the flow through the distributor plate will increase the pressure and therefore the push of the interface between oil/water is somewhat down. The flow of water that is pushed under the distributor plate is also clearly seen in the picture (see *Figure 5*). Relatively high velocities through the distributor plate are visible for both the forward and backward flow through the distributor plate area. High velocities are seen at the free surface between oil and water, especially in the vertical central plane of the separator.

In both the oil and the water zones – away from the central plane – backward flows are found in the numerical simulation.

Tracer Response and Hydraulic Efficiency Indicators in Flow-zones

Tracer Response in Oil-zone

The results from the tracer simulations performed in the separator are summarised and presented in *Table 1.* A theoretical retention time of 169.7s was expected for the oil. The calculated median retention time from the tracer response curve is 145.9s. The total short-circuiting number is one for ideal plug flow. The total short-circuiting number is 0.86. The value is reasonably close to unity, indicating little stagnant space in the oil flow. The tail of the tracer response curve indicates the mixing in the oil flow. The hydraulic indicator for the 'worst' short-circuiting states that values as low as 0.05 clearly indicate serious short-circuiting. The test separator has an indicator of 0.33. The dispersion index is between 2.5 and 3, which is the indicator for mixing in the oil flow, since a 'good settling basin' should have a dispersion index of less than two.

Tracer Response in Water-zone

The water flow through the separator has a theoretical retention time of 835.1s. The hydraulic indicators for the water presents non-favourable separation conditions for oil-drops (and small sand particles) in the separator. This number as well as the re-circulating pattern in the flow field, indicate high degree of mixing. The most serious short-circuiting is clearly indicated with a value as low as 0.07 – indicating that some flow is passing through the separator much faster than the theoretical retention time.

Figure 4: Sketch of the Test Separator

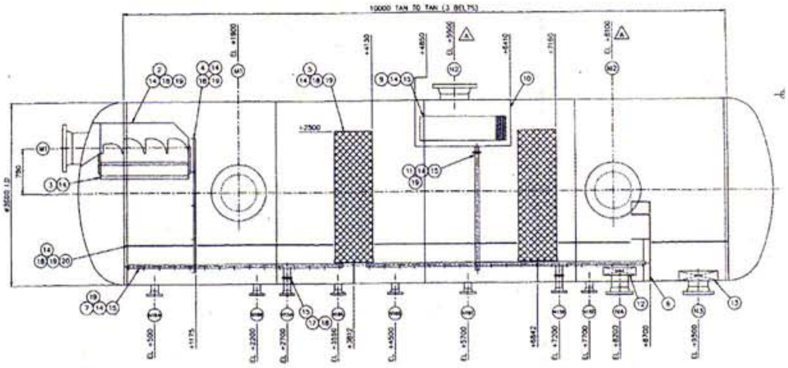
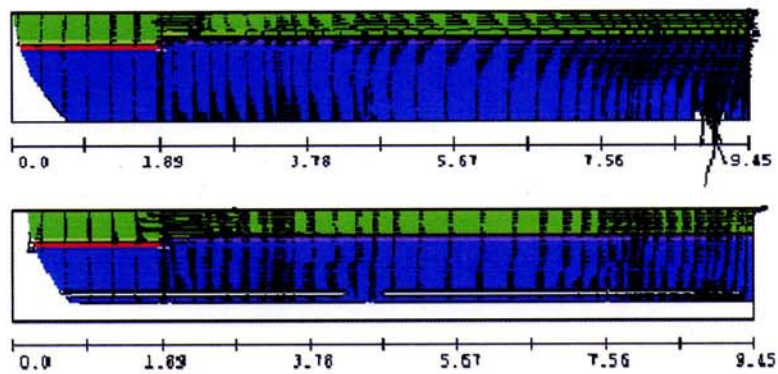


Figure 5: Numerical Simulation of Liquid Flow in the Test Separator



The flow patterns are seen in vertical slices, from inlet to weir, along flow direction. Maximum velocity is different in the two different views. Top view is along central plane and max velocity is 59.4cm/s. Lower view is about midway between central and wall plane and max velocity is 32.8cm/s.

Conclusions

Offshore separators are huge pressurised vessels, in which oil, water, gas and sand will be separated. The operation of the separator is to provide sufficient time to allow the phases to separate by gravity. The operation in the field and the internal devices should make efficient and low maintenance separators.

Advanced multiphase flow modelling by the CFD programme FLOW-3D can simulate the fluid dynamic effects in a gravity separator. The modelling of flow in separators is based on dividing the separator into different zones and limits the study to two phases at a time. By this method the entire separator can be analysed. FLOW-3D has been used in a number of studies and engineering jobs.

The rheology of oil/water emulsions is described mathematically. The bulk viscosity models generally lack validation towards relevant field or pilot scale model experiments. The drop-growth (coalescence rate) modelling implemented in a drift-flux flow model gives realistic behaviour of the dispersion

Table 1: Theoretical Retention Times in Oil and Water Flow Volumes for the Test Separator, Tracer Response Characteristics and Hydraulic Efficiency Indicators

Flow volume	***Prod. Rates***	***Theoret. retention time, Tr***	***Init appear. Ti***	***10% tracer T10***	***90% tracer T90***	***Total Short-. circ.***	***Most Serious short-circ.***	***Disper. index***
	M3/d	*s*	*S*	*s*	*s*			
Oil	*3,572*	*169.7*	*55.4*	*88.5*	*262.0*	*0.86*	*0.33*	*2.96*
Water	*1,479*	*835.1*	*62.5*	*153.6*	*1,909.6**	*0.67*	*0.07*	*12.43*

**T88 (88% of tracer particles counted at outlet).*

interfaces. Simulations of oil/water separation in a typical batch settler are performed.

The coalescence can be modelled with semi-empirical relations, but the relation between the coalescence rate and the mixing/turbulence in bulk flow zones in a separator is not yet sufficiently described.

CFD and numerical simulations represents a significant enhancement of design jobs and the great challenge is to combine the chemical and the fluid dynamics effects.

Many practical, design and redesign applications may be performed by CFD modelling and simulations. Examples of this are:

- development of vessel inlet configurations that improve the uniformity of the gas and liquid flows;
- sensitivity of a separator design to changes in operating conditions; and
- influence of internal equipment on separation performance.