

## PHENOMENOLOGICAL MODELLING AND SIMULATION OF FLUID FLOW AND SEPARATION BEHAVIOUR IN OFFSHORE GRAVITY SEPARATORS

Ernst W.M. Hansen  
SINTEF Energy Reserach  
Trondheim, NORWAY  
Email: ernst.hansen@energy.sintef.no

### ABSTRACT

The offshore activities, to produce oil and gas in the North Sea, have grown strongly during the last three decades. The equipment used for bulk separation operations is of considerable size and weight. The most widely used concept to separate bulk flows of water/oil/gas on platforms, are vessels in which gravity settling due to density differences of the fluids takes place.

This work discusses the distribution of the multi-phase fluid flow in a horizontal gravity separator. Computational Fluid Dynamics, (CFD) can provide valuable insight, and the fluid flow behaviour in the liquid bulk flow zone inside the separator is analysed. Water-in-oil forms unstable emulsions in processing and the drop-growth in the complex fluid will influence the separation efficiency. A phenomenological model of drop-drop collisions and coalescence is described and simulations performed.

zones (or regions).

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

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 the surge in the liquid flow rate.

Provide for the removal of solids that settle to the bottom of the separator.

Allow for variation in the flow rates of gas, oil and water into the separator without adversely affecting separation efficiency.

The simple design methods for gravity separators like those found in Refs. (Arnold and Stewart, 1986) and (Gordon and Fairhurst, 1987), do not match the complicated multiphase fluid flow behaviour through such equipment in any great detail. The design of the separators is not accurate or the design is too conservative. This frequently gives oversized vessels, both in volume and weight, and low separation efficiency, Ref. (Hancock and Hagen, 1986). Development of design and simulation tools, based on knowledge about basic hydrodynamic and physico-chemical principles, and possibly combining these effects, is under way, Refs. (Swanborn, 1988), (Hafskjold and Dodge, 1989), (Verlaan, 1990), (Hansen et al., 1991), (Hansen, 1993), and (Hansen et al., 1995). Even at the present stage, these tools represent a significant enhancement of the design and redesign

### INTRODUCTION

The separation process in petroleum production takes place in a process composed of a number of equipment units. In many fields, the process is composed of a number of separation units where the pressure is stepwise reduced, and gas flashed off. All oil/gas/water/sand separations are based on differences in density, and most separators are gravity separators, i.e., they utilize the common acceleration of gravity, and the separators are pressurized vessels. Later development has brought reliable high-g equipment to the market, particularly cyclones and centrifuges for oil and water polishing. The flow behaviour of a three-phase separator shows that different physical and chemical phenomena are important in different

jobs for gravity separators offshore, Refs. (Hansen et.al, 1993) and (Iversen, 1999). Computational fluid dynamics (CFD) applied to single-phase flow has become more and more important for modelling and design of industrial processes and its components, see Refs. (Boris, 1989), (Colenbrander, 1991), (Foumeny and Benyahia, 1993), (Hunt, 1991), and (Sharratt, 1990). The extension of single-phase CFD techniques to two- or multi-phase flow calculations is not a trivial task. Some of the problems and the difficulties have been reviewed by Ref. (Jayanti and Hewitt, 1991).

The present work discusses the distribution of the multi-phase fluid flow in a horizontal gravity separator. Computational Fluid Dynamics, (CFD) can provide valuable insight to develop and demonstrate the technology required for the realization of efficient, compact, low maintenance gas/oil/water separators. Water-in-oil forms unstable emulsions in processing and the drop-growth in the complex fluid will influence the separation efficiency. A phenomenological model of drop-drop collisions and coalescence are described and simulations performed. The fluid flow behaviour in the liquid bulk flow zone inside a separator is analysed. The use of CFD, demonstrates how the flow-field is related to inlet flow conditions and the performance of internals.

### FLOW BEHAVIOUR IN HORIZONTAL SEPARATORS

Each separator works at a fixed pressure. The multiphase fluid flow enters the vessel as a high momentum jet, hits a momentum breaker, and the liquid splashes into the liquid pool in the lower part of the vessel. The gas together with liquid drops 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 out coming gas and mist rises, and the oil and water will separate on the way to the outlet. In the upper part of the vessel, the oil and water particles together and 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 characterize the different flow regimes or zones, see Figure 1.

### FLUID SYSTEM AND SEPARATION MECHANISMS

The oil-gas-water-particle mixture which forms the inlet stream to a separator train, exhibits large variations in composition and physio-chemical properties. The thermodynamic properties, which determine the gas/oil ratio through the various steps of the process, are well known, and forms a well described system. The properties of the liquid phase, represented by bulk viscosity, emulsification, coalescence and settling properties, are less well understood, and consequently insufficiently described in terms of realistic mathematical models.

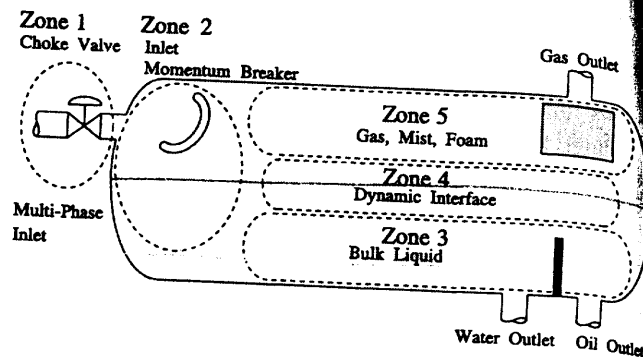


Figure 1. TYPICAL FLOW ZONES IN A HORIZONTAL GRAVITY SEPARATOR.

The separation process is dominated by two factors: (1) The emulsification process, which takes place in the choke and other equipment components with high shear, and (2) the coalescence and settling effects, where drops grow and settle or cream to its homo-phase.

### Dispersed Oil/Water Mixtures.

Normally the gas is released quickly in the first part of the separator, and the water-in-oil will form an emulsion in the lower part. The amount of water, the water-cut, and the stability of the emulsion will create the resistance in the flow-field. The bulk viscosity is an important parameter in the fluid dynamics and different correlations for apparent viscosity vs. water-cut for dispersions have been derived and presented in the literature. Some of the models suggested in the literature, Refs. (Pal and Rhodes, 1985), (Schramm, 1992), and (Pan et.al., 1995) are reviewed and plotted in Figure 2 for an oil/water mixture. The plot shows the relative viscosity vs. water-cut. The relative viscosity for emulsions is the viscosity for the emulsion divided by the viscosity of the continuous phase. All the models agree well for a dilute dispersion (water-cut below 10 %), but for higher water-cuts the models give quite a scattered result. Thus, the viscosity model used for fluid dynamic calculation in a gravity separator should rely on the emulsion behaviour investigated.

### Bottle Test (Batch Separation)

A common method of determining relative emulsion stability is a simple bottle test (a batch separation test), as described in reference (Schramm, 1992). A batch separation test involves agitating a mixture of oil and water to form a uniform dispersion, then removing the agitation to allow the dispersion to settle naturally. Batch sedimentation and coalescence experiments are an attractive method for study the separation behaviour of water-in-oil and oil-in-water emulsions because the experiments are simple and inexpensive to perform. When the percentage of water

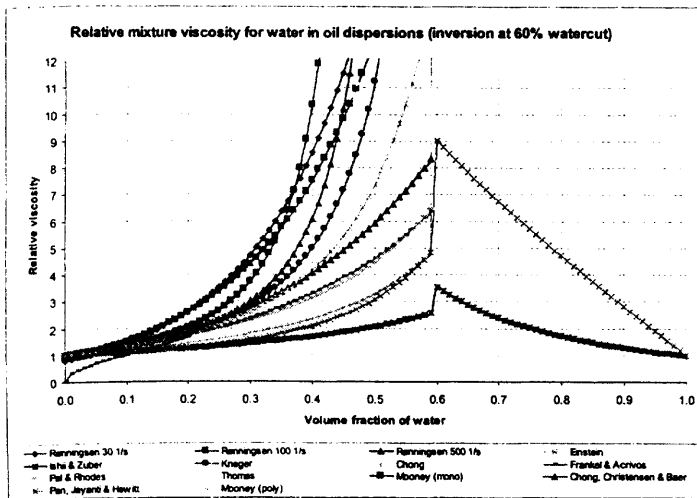


Figure 2. RELATIVE VISCOSITY CORRELATIONS FOR DISPERSED OIL/WATER MIXTURES.

in the mixture exceeds about 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 (in which a small amount of dispersed water may remain). The other is a "coalescence" interface between the dispersion and the bulk water phase (which contains a small part of dispersed oil). 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 advanced mixture model, such as the drift-flux model, see Refs. (Hansen, 1993), (Hirt et al., 1979), (Ishii, 1975), (Ungarish, 1993), and (Zuber and Findley, 1965). The drift-flux model is one of the multi-phase flow models in the commercial CFD program FLOW-3D, see Ref. (FLOW-3D).

The settling of drops gives rise to upward flow in the settler and the dynamical behaviour is well defined with the drift-flux model. The model regards the whole mixture as a single flowing continuum, by solving the volume continuity and the momentum equations for the whole mixture, and the mixture has macroscopic properties like the bulk viscosity, previously discussed. The model describes the relative flow of the immiscible fluids with different densities.

One of the important variables in focus is the local, instantaneous volume fraction - or water-cut. Furthermore, a drop-size distribution forms the macroscopic value, water-cut, in a poly-dispersed water-in-oil emulsion. In the simulation of the phases, the process is dominated by the coalescence and the settling/creaming rates of the dispersed phase. The settling rate depends on a number of parameters: drop-size

distribution, drop density, shear/turbulence and interfacial properties. The settling/creaming rates are determined by the density difference, and the density of the dispersion, computed by Stoke's law or models for hindered settling. The binary coalescence rate is the product of collision frequency and coalescence efficiency, i.e. the number of collisions that lead to a successful merging of drops, which gives the drop-growth in the mixture system. The driving forces in collision frequency are: the settling of drops, gradient flow and turbulence in the flow-field.

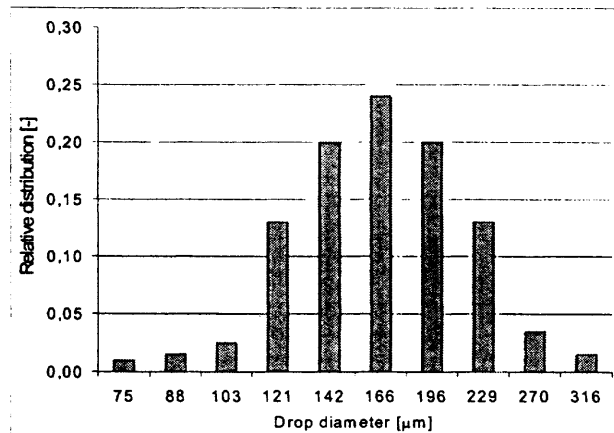


Figure 3. DROP-SIZE DISTRIBUTION FOR AN OIL/WATER MIXTURE.

### Modelling and simulation of batch separation

The mathematical modelling of coalescence in unstable oil-in-water emulsions is described in Refs. (Davis, 1985), (Hafskjold et al., 1991), (Hafskjold et al., 1997) and (Meijis and Mitchell, 1974), and more fundamental theories in Refs. (Coualoglou and Tavlarides, 1977) and (Randolph and Larson, 1988). The work presented in this context is part of the ongoing project EMU-SIM, see Ref. (EMU-SIM, 1999-2002). The project is a study to improve calculation-/design-tools for gravity separators. The coalescence model is further developed in this project and implemented in the commercial CFD program FLOW-3D.

Numerical simulations for batch settling of a poly-dispersed water-in-oil emulsion are performed and the results is presented in Figure 4. The water-cut in the emulsion was 20 %, the density difference of the two fluids is  $200 \text{ kg/m}^3$  and the height of the settler was 0.3 m. The drop-size distribution, see Figure 3, was initially configured in ten different drop-classes. The Sautern mean diameter in the distribution is  $160 \mu\text{m}$ . Settling in the batch was simulated with and without a coalescence model, and as seen in Figure 4 the separation run faster with the drop-growth

model (coalescence model) included. The separation time for the present batch system is 430 seconds, calculated by Stoke's unhindered settling equation.

The separation time is 1300 seconds calculated by the hindered settling equation from Kumar and Hartland, 1985, see Ref. (Panoussopoulos, 1998). The simulated result with a constant Sautern mean drop-size  $160 \mu\text{m}$  (original drift-flux model) gives a long settling time, about 1800 seconds. The simulated result with a drop-growth model (coalescence model) shows approach of the two interfaces after 1150 seconds, which is in good agreement with the hindered settling equation. The difference in the simulated results are presented in Figure 4.

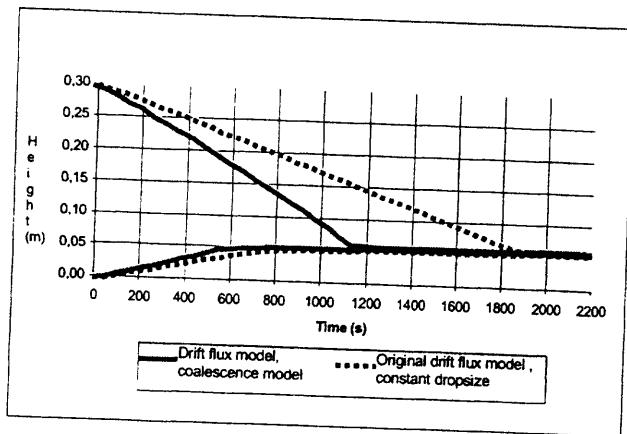


Figure 4. SIMULATED RESULTS OF SEPARATION INTERFACES IN BATCH SETTLING.

## FLUID FLOW BEHAVIOUR IN A HORIZONTAL SEPARATOR

Gas/liquid and liquid/liquid systems forms separated flow systems, due to density differences, in pipelines or separators. Between the two fluids a dynamical interface will form, stratified or annular in pipelines or stratified in separators. The characteristics of such flow systems may be developing flows, entrance flows, co-current flow transition, etc and the full dynamical interaction between the phases are important.

The gas and liquid flows with rather high velocities enter through the pipe and the inlet device to the separator. Inlet devices, such as cup-shaped plates, turbine-vane arrangements or cyclone-arrangements are normally mounted as a momentum breaker device. The momentum breaker device will lower the flow velocities and separate the gas and the liquid phases quickly with a minimum space required in the longitudinal direction. The flow off the momentum breaker will further intro-

duce the gas (gas and mist) and liquid phases (water and oil) into the gas volume or the liquid pool of the separator, respectively and further utilize the full three-dimensional vessel. A horizontal three-phase separator is normally about half-filled with liquid (oil/water) to allow the gas to evacuate with a proper retention time in the upper part of the separator.

To model and simulate a high momentum multi-phase flow in the complicated inlet momentum breaker region, which introduce the flows into the three-dimensional gas and liquid bulk flow zones, are beyond the capability of the current CFD programs. The fluid flows off the momentum breaker device have to be estimated and introduced as an inlet flow condition to the bulk flow zones in a horizontal separator.

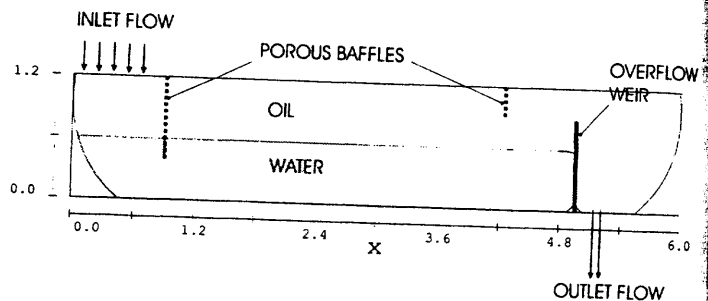


Figure 5. INITIAL OIL/WATER CONFIGURATION IN THE LIQUID FILLED PART OF A HORIZONTAL SEPARATOR.

## Fluid flow modelling and simulation

CFD simulations perform the study of the fluid flow behaviour in the liquid bulk flows in a horizontal separator. The separator is 6.0 m long and the diameter is 1.95 m. The initial flow and fluid configuration are given in Figure 5, presented in a longitudinal vertical slice through the separator, the liquid level (oil/gas interface) is 1.2 m. The inlet and outlet flow, the porous baffles, the overflow weir and the oil/water interface are shown in the figure. In most three-phase separators a distributor plate, the porous baffle separating the inlet zone and the bulk flow and separation zones, has a strong impact on the flow pattern in the oil- and water zones. The distributor plate has a low porosity (fraction of area that is open to flow) and thus a pressure loss for the flow through. The distributor plate is not extended to the bottom of the vessel and it is important to introduce the water part into a proper simulation model of the liquid filled part of the separator. Figure 6 presents the inlet flow condition in a cross-section

