OTC 16137

Field Confirmation of CFD Design for FPSO-mounted Separator

Chang-Ming Lee / NATCO Group, Eric van Dijk / SBM-IMODCO, Inc., Mark Legg / SBM-IMODCO, Inc., John Byeseda / NATCO Group

Copyright 2004, Offshore Technology Conference

This paper was prepared for presentation at the Offshore Technology Conference held in Houston, Texas, U.S.A., 3–6 May 2004.

This paper was selected for presentation by an OTC Program Committee following review of information contained in an abstract submitted by the author(s). Contents of the paper, as presented, have not been reviewed by the Offshore Technology Conference and are subject to correction by the author(s). The material, as presented, does not necessarily reflect any position of the Offshore Technology Conference or its officers. Electronic reproduction, distribution, or storage of any part of this paper for commercial purposes without the written consent of the Offshore Technology Conference is prohibited. Permission to reproduce in print is restricted to an abstract of not more than 300 words; illustrations may not be copied. The abstract must contain conspicuous acknowledgment of where and by whom the paper was presented.

Abstract

Two new Free Water Knockout (FWKO) vessels were installed on an FPSO facility operating offshore Cabinda, Angola. They were installed to accommodate larger slugs and to debottleneck downstream facilities to cope with increasing water production. The paper describes process design and CFD work for placement of internal baffles to mitigate fluid sloshing in the new separators. Three CFD simulations were performed. First we demonstrated that the separator would not operate properly without custom-designed internal baffles. The second simulation demonstrated that customized baffles in the new vessel could suppress sloshing and prevent water spill over into the oil discharge stream. The final simulation showed that the advanced baffle arrangement would control the problems due to sloshing when fluid flow was introduced into the simulation. Field data is presented to confirm the validity of the simulation work and successful operation of the separators. The work gives confidence in the use of CFD to design separators with minimum weight and size. We show that for critical applications, the CFD design should be applied to the particular case – baffle designs are not "one-size-fits-all".

Introduction

The purpose of this CFD (Computational Fluid Dynamics) study was to evaluate the design and performance of internals installed in a horizontal Free Water Knockout separator (FWKO), located on SBM's Kuito Floating Production Storage and Offloading (FPSO) facility in Kuito Field Block 14, offshore Cabinda, Angola. The new FWKO design included both a proprietary cyclonic inlet device and perforated baffles for flow distribution. The new internals were installed to accommodate larger slugs and to debottleneck downstream facilities to cope with increasing water production.

The internal baffles were evaluated for suppression of fluid sloshing inside the vessel when it is subjected to acceleration wave motion under different sea states. Cyclonic inlet devices are industry proven separation technology for reducing or eliminating foam in a separator and for improving the efficiency of separation. Since the primary goal of the simulation work was to design perforated baffles that would allow the separator to operate effectively at the maximum wave-induced motion on the FPSO, the cyclonic inlet device was modeled in the CFD simulation as a simplified block component.

CFD Background

Computational Fluid Dynamics (CFD) is a mathematical technique used for determining the numerical solution of the governing equations of fluid flows (e.g., the unsteady "Navier-Stokes Equations" for Newtonian fluid dynamics) while sequencing the solution through space and time in order to generate a numerical description of the dynamic flow field. CFD can not only predict the behavior of fluid flow, but also the transfer of heat, mass (e.g., precipitation, dissolution), phase change (e.g., melting, freezing, boiling), chemical reaction (e.g., combustion), mechanical movement (e.g., motion of pistons, fans, rudders and impellers), and stresses or deformation of related solid structures (e.g., a mast bending in the wind).

The continued rapid improvement in the speed of computer hardware and increased size of available memory have led to the emergence of CFD since the 1960's, and the inception of the commercial CFD software industry started in the early 1980's. CFD complements theoretical and experimental fluid dynamics by providing a flexible and cost-effective means of testing theoretical advances or simulating the performance of alternative configurations for conditions that are very difficult to physically test in a complex flow system on an experimental basis. The use of CFD for the study of turbulent flows has become so prevalent that it may be viewed as a new dimension of fluid dynamics along with pure theory and pure experiment.

A CFD simulation can provide enhanced fluid flow visualization that would not otherwise be available, allowing CFD users to readily gain a better understanding of vessel designs and processes. Use of CFD simulation can lead to shorter design cycles and further compress the time between the conceptual stage of a project and field implementation. In addition, CFD studies can quickly guide engineers to the root of performance problems. This is advantageous in the diagnosis and troubleshooting of existing equipment, evaluating retrofit designs, and minimizing equipment down time. Most importantly, it is apparent that process optimization will result in substantial saving of time and expense.

Process and Vessel Design

V-1301B is a horizontal free water knockout separator. The vessel is 4267 mm diameter with a seam-to-seam length of 12.19 m (excluding the vessel heads). The normal oil level is 2388 mm and the normal water level is 1016 mm (both estimated from B.O.V.) Figure 1 shows a two-dimensional drawing and a three-dimensional sketch of the V-1301B vessel.

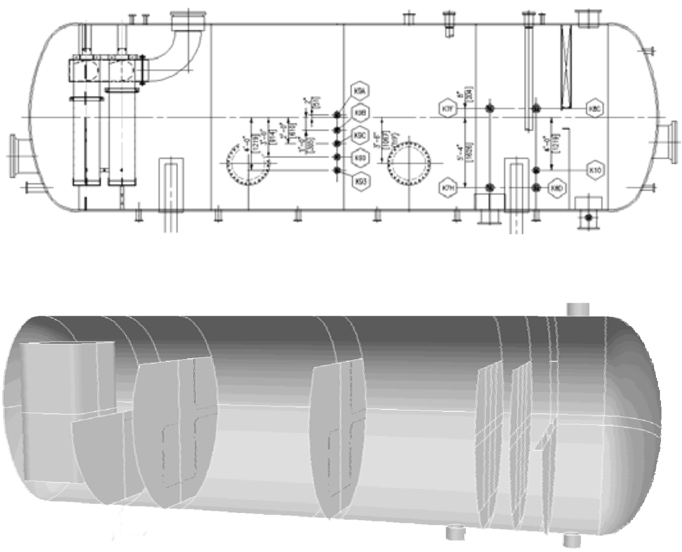


Figure 1. 2D Drawing and 3D Model of the FWKO Vessel.

The new process vessel is designed to continue operation while accepting 25 m^3 slugs of either oil or water. The process design includes the required slug volume between the NLL and LAH in each of the phases. Two limiting cases exist for the process design: maximum water flow and maximum oil flow. The maximum water flow case was used for the simulation work because the critical design issue was to prevent water from spilling over the oil weir after the vessel received the 25 m^3 slug of water.

The operating parameters for the CFD study are:

Temperature	50 °C
Pressure	690 kPa(g)
Gas Flow Rate	55 MMSCFD
Oil Flow Rate	22,000 BPD
Water Flow Rate	74,250 BPD
Normal Oil Level	2388 mm (from B.O.V.)
Water Level High	1702 mm (from B.O.V.)

The density and viscosity of the wet hydrocarbon gas phase are 5.7 kg/m^3 and 0.012 cp. These values are based on gas compressibility factor of 0.99 and at the specified operating conditions of 690 kPa(g) and 50 °C. Under the same operating conditions, the density of the crude oil is 907.0 kg/m^3 and its viscosity is 42.0 cp, whereas, the density and viscosity of the produced water is 1026.0 kg/m^3 and 0.9 cp.

Equipment Installation Details

The Kuito Field is an oil field located in Block 14-1, approximately 373 m of water depth, about 75 km offshore Cabinda, Angola. SBM's Kuito tanker had been converted into a Floating Production Storage and Offloading (FPSO) facility with oil processing, gas processing, water injection and all support systems in project Phase 1A and 1B. Project Phase 1C included tie back of additional subsea manifold and increased water handling from 20,000 to 45,000 BPD. Project Phase 2A increased gas compression, gas treatment and water handling from 45,000 to 135,000 BPD. The full range of wave motions are experienced due to sea states including significant surge, heave, sway, pitch, roll and yaw. These wave motions will adversely affect the topside facility's equipment and system operation. Adding internal baffles has proven to be a practical means to effectively mitigate the liquid sloshing in the vessel and to provide the improved operating performance of the equipment under consideration.

The V-1301B FWKO is located on the starboard side close to vessel bow on the upper deck. Because of the relatively large diameter of the FWKO vessel, the placement of internal baffles is critical to water spillover control and for controlling fluid motion inside the vessel.

The proposed location of V-1301B FWKO vessel is on the starboard side of the upper deck Figure 2 illustrates the perspective view of the physical position of the vessel on the lower deck in reference to the FPSO's hull and keel plate.

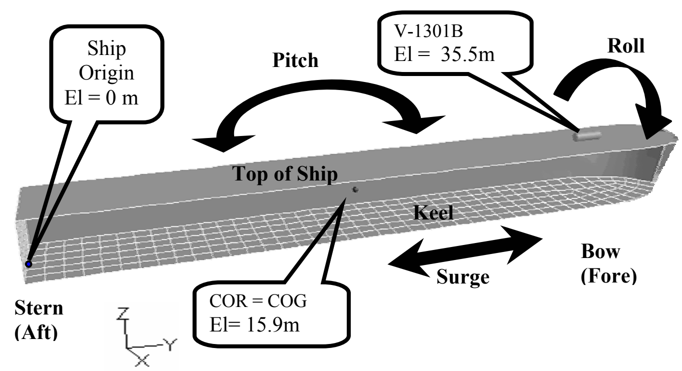


Figure 2. Layout of V-1301B FWKO on SBM's FPSO Upper Deck.

The origin of the computational domain was estimated at 16.19 m off the centerline of the vessel, 95.50 m forward from the origin, and 19.53 m above the COG (equivalent to elevation of 35.47 m above the keel plate). The longitudinal axis (Y-direction) of the FWKO center is parallel to the FPSO's fore and aft directions and the vertical axis (Z-direction) is consistent with the heave motions direction.

The vessel center of rotation (COR) is assumed to be the same as the vessel center of gravity (COG). It is usually subjected to the influence of many parameters including the riser configuration, payload on the topsides, frequency and intensity of the sea state, and the frequencies of all six-degree wave motions. We assumed that the COR remained the same position and did not vary with sea state conditions. The relative distance from COG with respect to the V-1301B's mathematical origin is estimated at values of X = 16.19 m, Y = 106.88 m, and Z = 19.53 m.

Case Studies

There were a total of three cases in this study. These studies were for one operating level at operating conditions of one pressure and temperature (690 kPa(g) and 50 °C) and for three-phase flows in the V-1301B FWKO vessel.

The cyclonic inlet device was simplified as one large rectangular-shaped solid block. The position and configuration the cyclonic inlet device was the same for each case.

Fluid flow was evaluated in the last case (Case #3).

Case 1: Vessel with four perforated plates located as shown on Figure 3.
Case 2: Vessel with modified internals design. An additional "combo plate" was added near the final weir plate, and a lip was added to the weir plate to control roll induced sloshing.
Case 3: All internals were the same as in Case #2 but with fluid flow introduced into the V-1301B vessel. Detailed boundary conditions are described in a subsequent section.

Sea State for Simulations. The selected sea state for the CFD simulations was defined according to the ten-year extreme storm condition. The details are listed in Table 1.

Several assumptions were made regarding the sea state in this CFD study. First, only pitch and roll motions were taken into account and zero phase angles were assumed between all of these wave motions. Second, the averaged wave motion periods of the ten-year extreme sea state was 15.0 seconds (the actual period is between 14.0 to 16.0 seconds according to SBM's FPSO General Conditions & Data Specification). Finally, a single period was exclusively applied to all wave motions in the specified sea state and no phase angle was included between the wave motions.

Table 1. Wave Motion Parameters of 10-Year Extreme Condition.

Sea State Parameters	Units	Values
Roll (single amplitude)	degree	4.35
Pitch (single amplitude	degree	1.40
Significant Wave Height	meter	3.81
Tide	meter	1.22
Return Period	second	14.0-16.0

Simulation Results. Internal wave-induced motion suppression baffles investigated in this study included solid, perforated, and combination plates. Two-dimensional contours of oil volume fraction in the cutting-view plane (ZY-plane at the vessel longitudinal center) were generated during the transient simulation process to present the dynamic variations within the entire flow domain. The last case (Case #3) included the combined effects of wave motions plus fluid flows (both oil and water phases) through the vessel under design conditions.

Vessel Internals. The primary goal of this CFD study was to evaluate the design of internals for V-1301B FWKO vessel to suppress the sloshing of liquid inside the vessel when it is subjected to acceleration wave motions under the 10 year sea state on the upper deck of the FPSO. Internal wave-induced motion suppression baffles investigated in this study included solid, perforated and combination plates.

A weir plate (at an elevation of 1113 mm calculated from the bottom of the vessel shell) might also be considered as an element of the internals. Although its position was fixed, the configuration of the weir was modified according to the results obtained from CFD Study Case #1 in order to achieve the goal of preventing water phase spillover. All of the internal materials (perforated plate baffles, combo plate and solid weir) used in this study were defined as either porous media or solid wall boundary conditions in the CFD simulation.

The perforated plates had a thickness of 9.53 mm with total orifice open area that varied according to the position and purpose of the particular baffle. The baffle positions were at 1829, 3200, 6858, 9957, and 10566 mm, respectively (calculated from the head and shell seam of the FWKO inlet). The design heights of the perforated plates (plate #2 to plate #5) were all 914 mm above Oil Normal Liquid Level (NLL), except the height of the plate #1 was 584 mm above Water Liquid High (which was at 1549 mm from B.O.V. or right at the horizontal centerline of the FWKO). All perforated baffles had a 152 mm gap between their undersides and the bottom of the vessel in order to allow space for accumulation of solids.

Figures 3 and 4 illustrate the full 3D cutaway views (drawn to scale) of the cylindrical V-1301B vessel including all of the internals used in the CFD studies. Case 1 used baffles 1 through 4. Cases 2 and 3 used baffles 1 through 5.

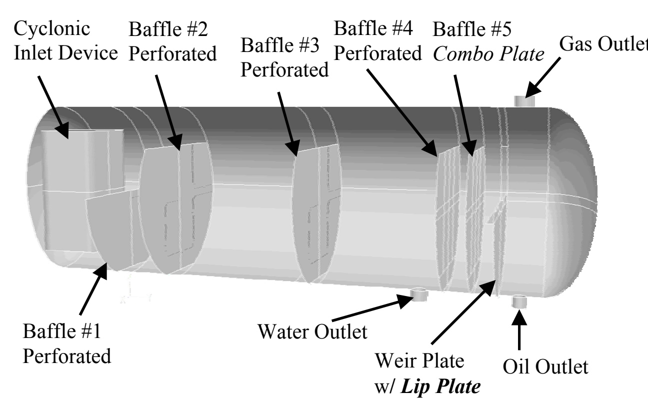
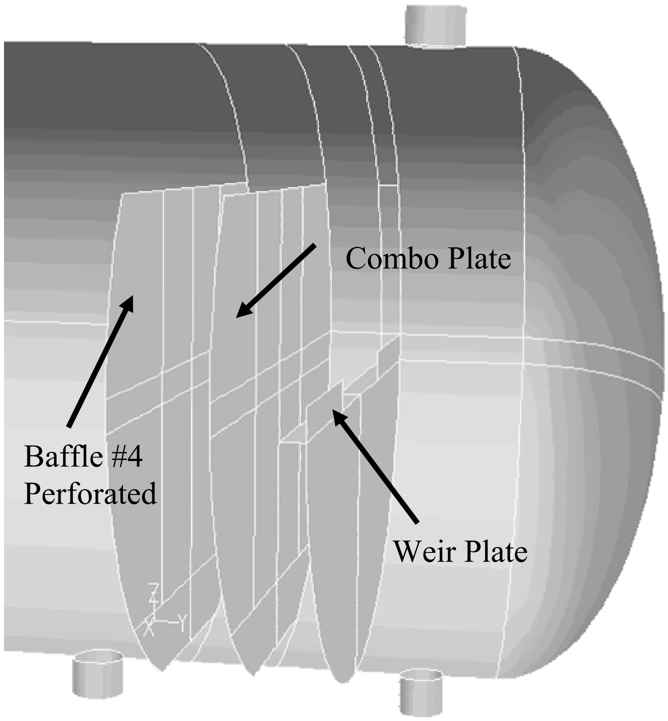


Figure 3. 3D Perspective View of V-1301B for the CFD Study



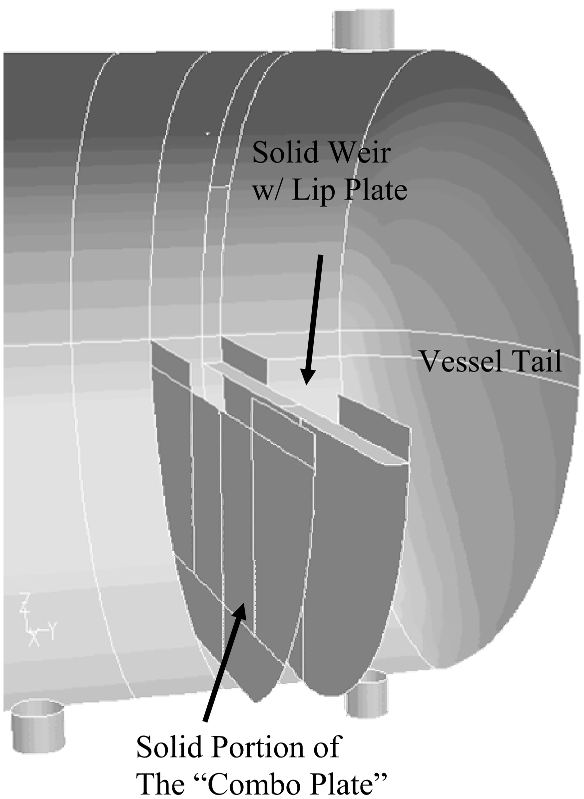


Figure 4. Detailed View of V-1301B's Internal Baffles near The Liquid Outlet Region

This study included Cases #1 to #3. The first two cases were run at "no flow" conditions to simplify computation based on our experience that fluid flow has only a small but beneficial effect on sloshing motion of interfaces between phases. The simulations were run in excess of three wave motion periods to ensure the results were representative. The last case (Case #3) involved the study of the combined effects of wave motions plus fluid flows (both oil and water phases) through the vessel under design conditions.

Two-dimensional contours of volume fraction of the oil phase on the cutting-view plane (i.e. ZY-plane) located near the vessel's longitudinal center were created during the transient simulation to exhibit the dynamic fluid movements (resulting from the wave motion effects) within the entire flow field. As can be seen from the two dimensional contours of oil volume fraction in Figures 5 and 6, with the original predesigned internal baffles, there was water spillover behind the weir at simulation time of 24.0 seconds. This was primarily from the relatively intense roll motion experienced by V-1301B and the presence of a wave crest due to the pitch and roll motion of the vessel.

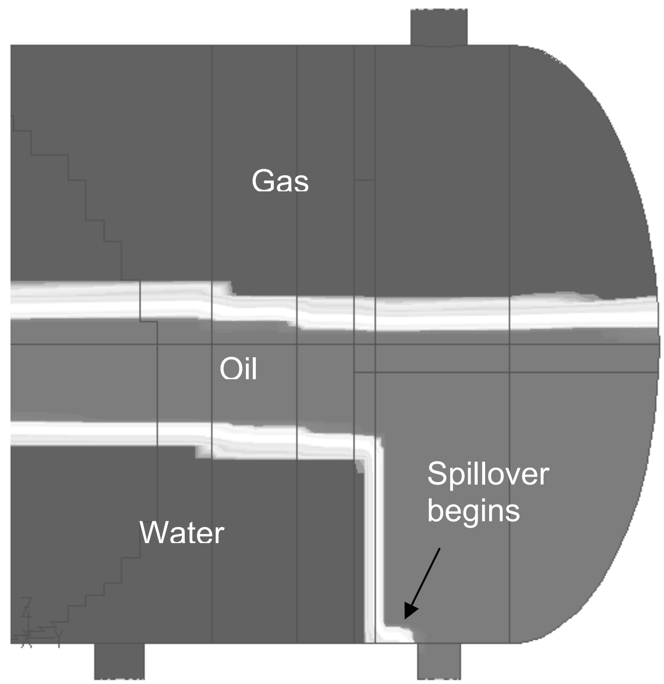


Figure 5. Contour of Oil Volume Fraction near Vessel Center Cutting Plane.(For Case #1 at Simulation Time = 24.0 seconds)

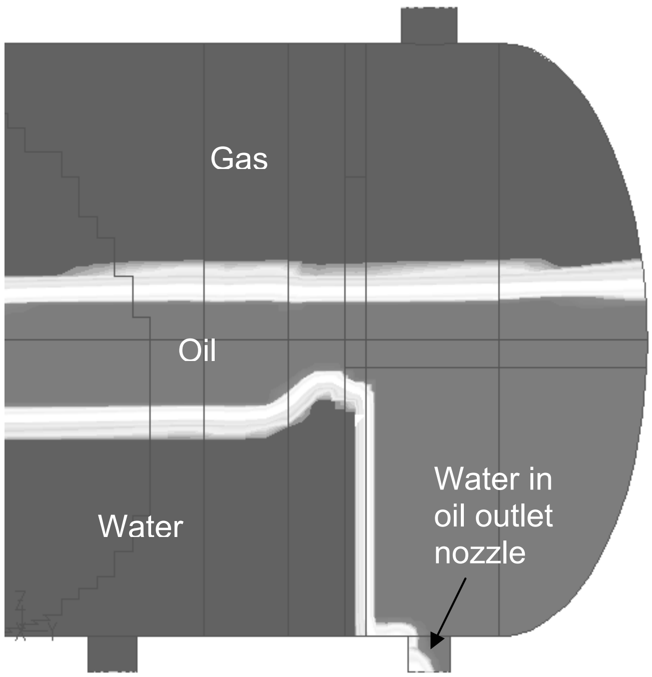


Figure 6. Contour of Oil Volume Fraction near Vessel Center Cutting Plane. (For Case #1 at Simulation Time = 28.0 seconds)

To eliminate water spill over the weir into the oil compartment, we located the last baffle (Baffle #5) (comprised of both solid portion and perforated portions) near the oil weir and decreased the open area on both Baffles #4 and #5. With these changes, the water/oil interface was well controlled and there was no water spilling over the oil weir (as illustrated in Figure 7 at simulation time of 44.4 seconds).

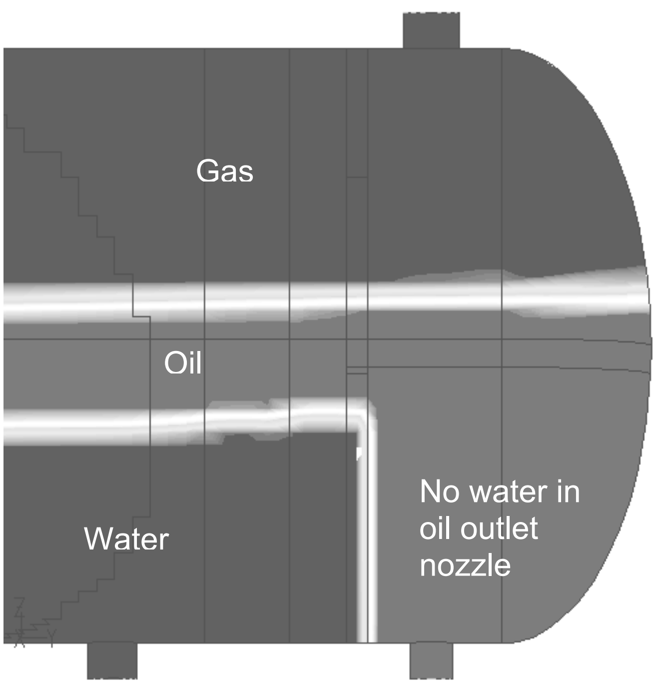


Figure 7. Contour of Oil Volume Fraction near Vessel Center Cutting Plane. (for Case #2 at simulation time = 44.4 seconds)

In order to visualize the roll motion effect within the vessel, an animated three-dimensional perspective view showing the deformation of water/oil interface was created for Case #2. Figure 8 shows a snapshot of an individual frame from the animation sequence (at simulation time of 44.4 seconds).

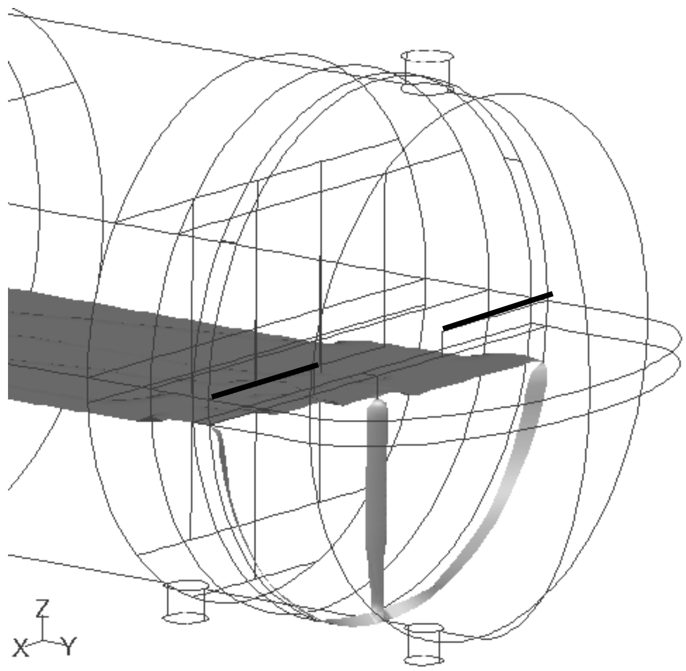


Figure 8. Plot of 3-D Water/Oil Interface Tracking in Case #2 (at simulation time = 44.4 seconds).

In the last case study (Case #3), the internal configuration was the same as for Case #2, except that flow of both oil and water through the vessel was introduced into the calculation. Gas flow was ignored to simplify the simulation since there would be minimum impact from gas phase flow to the deformation of the water/oil interface. Figure 9 shows a snapshot from the animation sequence for Case #3. It shows that introduction of both liquid streams into the flow domain introduces no adverse effects into the calculated results. The baffle system maintains control of the oil-water interface and no spill-over of water into the oil outlet was observed.

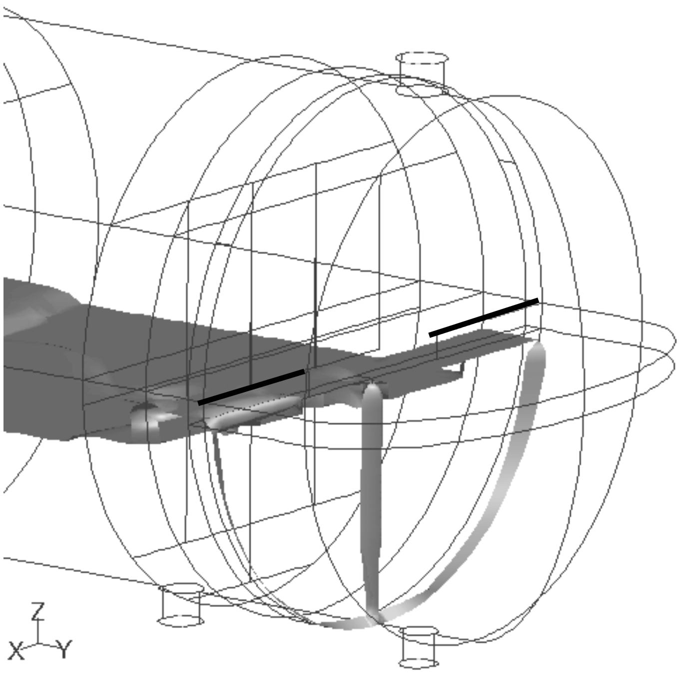


Figure 9. Plot of 3-D Water/Oil Interface Tracking in Case #3. (at simulation time = 31.2 seconds.

Observations from Field Operation

The FWKO vessels were taken into production in March 2003. The vessels have been operating continuously since start-up at about 75% of design throughput and an inlet water content in the order of 40 to 60 volume %. The API of the crude produced is in the 20° range, making it a heavy oil. One of the complicating aspects of the Kuito crude is the formation of calcium napthenate scale that affects separation performance negatively. Although the crude properties do not favor oil water separation, good process performance has been observed from the start. The oil content of the water outlet has been typically in the 200 ppmv range, which is very low compared to the 2000 ppmv design maximum. The design maximum BS&W of the oil outlet is 10 volume % and in practice figures below 5 volume % have been observed most of the time. It has to be mentioned, though, that 10 year storm conditions have not been encountered, the weather conditions at site are typically benign. Overall, the performance of the FWKO vessels so far seems to justify the vessel internal design.

Summary and Conclusions

Three CFD simulations were performed. The first two simulations were done without flow to simplify modeling. First we demonstrated that the separator would not operate properly without custom-designed internal baffles. Water would spill over the final weir. The second simulation demonstrated that the customized baffles in the new vessel could suppress sloshing and contain the oil-water interface so that water would not continuously flow out of the vessel with the oil discharge stream. The final simulation (with flow) indicated that the advanced baffle arrangement would control the problems due to water spilling over the oil weir. Field data is presented to confirm the validity of the simulation work.

Acknowledgements

We wish to extend our appreciation to NATCO Group and SBM-IMODCO management for permission to publish this paper. We also appreciate the support of SBM-IMODCO for giving operational feedback from the field. Our thanks also go to Ted Frankiewicz for providing editing and material support.