Novel vertical flotation cell succeeds for deepwater floating structures

Using a combination of CFD and physical testing, a flotation cell was developed for use on floating and fixed platforms. The technology has been operating successfully in the Gulf of Mexico for more than 18 months.

Dr. Ted Frankiewicz, Dr. Chang-Ming Lee and **Kevin A. Juniel**, NATCO Group, Inc.

Operating companies continue moving into ever-increasing water depths to produce oil and gas reserves. Producing in these deepwater locations has caused a shift from the traditional fixed-leg platforms to floating structures, such as tension-leg platforms (TLPs), Spar platforms, and Floating Production, Storage, and Offloading (FPSO) vessels.

This shift to floating structures has created new challenges for the designers of separation facilities. For example, due to the high cost required to construct floating platforms, it is imperative that the size and weight of processing equipment be minimized. In addition, the fluids in processing equipment that is installed on the topsides of these floating structures are subject to wave-induced motions that can at times be quite severe.

Generally speaking, separation facilities, including water treatment equipment, are expected to continue operating, even in heavy seas with vessel pitches of up to 4° and vessel rolls that can exceed 10°. These issues must be taken into consideration when designing topsides equipment, to minimize any adverse impacts to capacity and performance.

To ensure that they reliably meet their overboard water discharge quality requirements, deepwater operators have adopted a more-or-less standard approach of cleaning produced water by using liquid-liquid hydrocyclones followed by induced gas flotation (IGF). To conserve space, vertical column flotation is the preferred configuration. There are, however, special challenges in designing a suitable column flotation unit for service on a floating platform. To address these challenges, NATCO assembled a team of experienced engineers to evaluate the designs of existing technologies. They then developed a new design that would be optimal for installation on floating platforms.

Commercially available designs were reviewed as the starting point for development of the vertical column flotation (VCF) technology that came to be known as VersaFlo™. The novel development and commercial deployment of this technology was recognized by an OTC 2004 Spotlight on Technology Award. The technology has patents pending in the US and in appropriate foreign countries.

In addition to reviewing existing technology, computational fluid dynamics (CFD) simulations were used to identify key fluid flow issues for optimizing the performance of column flotation. These issues include:

- Minimizing net downward water flow velocity
- Provision of coalescence assistance to increase oily contaminant particle size
- Uniform distribution of inlet water over the cross-section of the cell
- Providing a uniform, reliable distribution of flotation gas
- Minimizing the size of gas bubbles used for flotation
- Minimizing internal recirculation zones that can by-pass oily water around swarms of rising gas bubbles
- Elimination of dissolved gas break-out slugs to prevent the upsetting of the oil skimming process
- Control of the oil/water skimming in the flotation cell during sloshing induced by movement of the host platform
- Elimination of short-circuiting to the outlet nozzle.

A suite of additional CFD simulations was conducted to define the means required to reduce or eliminate the negative impact of the above, listed factors. Several series of physical tests were conducted during the course of VersaFlo™ development to verify the predictions of the CFD simulations, and to test the performance of proposed solutions to the identified problems.

PHYSICAL TESTING

The physical test program was carried out at the NATCO-Axsia test facility in Gloucester, UK. The primary test tank was 60 in. in diameter and 15 ft from top to bottom, Fig. 1.

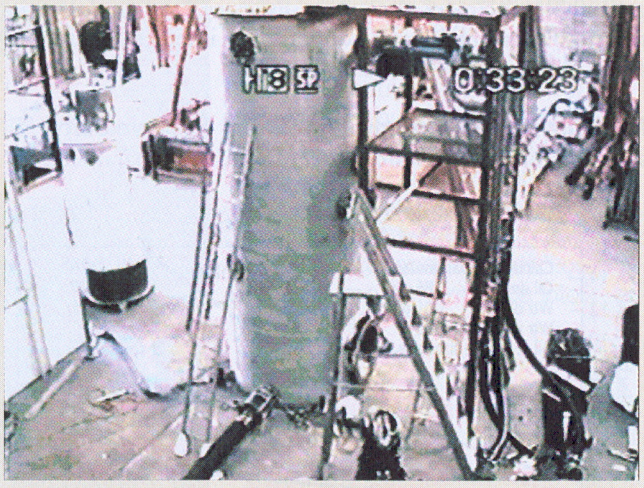

Fig. 1. The 5-ft diameter by 15-ft tall test tank at the NATCO-Axsia facility in Gloucester, UK, that was used to validate and extend the results of CFD simulations for the VCF technology development program.

Water was recirculated from the tank's bottom outlet to the tank inlet at rates up to 10,000 bwpd. Separately, water could be recirculated to various development models of the gas eductor at rates up to 5,000 bwpd. Water used in the test program included 2% dissolved salts (NaCl) to provide for liquid viscosity, liquid density and a gas bubble size distribution that would be more representative of what would be experienced in actual operation.

As mentioned, the CFD simulations identified one key performance parameter to be, the need to uniformly introduce and distribute gas bubbles of suitable size for effective induced gas flotation (IGF). Accordingly, a parallel development effort was instituted to develop the required radial eductor. This development, although key to the successful development of VCF, is the subject of on-going patent prosecution and beyond the scope of this article.

GAS FLOTATION MECHANISMS

Studies of induced gas flotation have defined three mechanisms for removing oily contaminants from water. These include:

- The coating of gas bubbles with oil films
- The attachment of the oil droplet or oil-coated solid to the gas bubble
- The hydraulic drag of contaminants in the wake of a rising gas bubble.

Contaminant attachment to gas bubbles, because it is weak, is temporary, and a contaminant may require the assistance of several gas bubbles before it reaches the water surface from where it can be skimmed. This mechanism is likely to be most effective for smaller gas bubbles and smaller contaminants.

The third mechanism, hydraulic drag, is thought by many to be the primary mechanism operable in induced gas flotation. With this mechanism, a buoyant particle is carried in the wake of a gas bubble toward the water's surface. Again, the effect is weak and a contaminant will require interaction with several gas bubbles before it is successfully carried to the water surface.

DESIGN FEATURES

It is clear that all of the aforementioned flotation mechanisms will be more effective as the number of interactions with gas bubbles increases. As will be discussed further, the opportunity for gas bubble and contaminant interactions increases as gas bubble diameter decreases.

Minimizing net downward water flow velocity. The selection of a downward water flow design parameter is a compromise between the need to minimize the flotation cell's cross-sectional area while also minimizing downward velocity against which a gas bubble and contaminant must rise. For VCF, a downward velocity of 2.0 ft/min. was selected. At this rate, gas bubbles larger than 125 microns will rise under common field conditions, Fig. 2. As will be discussed below, 125-micron gas bubbles are effective for IGF.

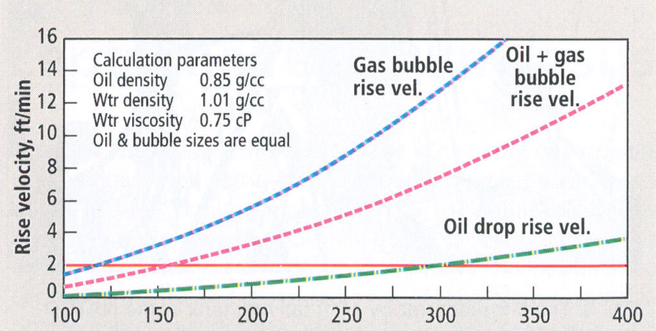

Fig. 2. Rise velocity for particles in a flotation cell.

Provision of coalescence assistance to increase oily contaminant particle size. The VCF was designed to provide two stages of oily contaminant coalescence and/or flocculation. The first is in the cyclonic inlet, where 15 to 30 Gs of force are generated, inducing contaminants and large gas bubbles to migrate to the center of the inlet. As the localized concentration of contaminants increases, so does the opportunity for them to grow in size. The cyclonic inlet device used in VCF is based on gas-liquid cylindrical cyclone (GLCC) technology.

The second opportunity for contaminant coalescence is provided by NATCO-lescer media. This media is installed below the inlet device and provides a large non-contiguous surface area on which oil droplet coalescence can occur. Because the surfaces are non-contiguous, plugging potential for the media is minimized. Fig. 3 shows NATCO-lescer J-VSP type media installed in a 40,000-bwpd VCF system currently in service in the Gulf of Mexico.

Uniform distribution of inlet water over the cell's cross-section and elimination of dissolved gas breakout slugs to prevent the upsetting of the oil skimming. During the development of VCF, two types of produced water inlet designs were considered: pipe distributor and cyclonic. The pipe distributor suffers from two basic problems in this type of application. First, it does not permit for disengagement of gas slugs that are often introduced to a flotation cell that is downstream of a higher-pressure separator, either directly or through a deoiling hydrocyclone.

Although gas solubility varies with temperature, it is not uncommon for the fluid entering a flotation cell or skimmer to have a gas volume fraction (GVF) of 25% to 50%. At these high GVFs, the gas bubbles evolving from produced water can gather in interconnecting piping to form substantial slugs that can upset both vessel level control and oil skimming in a single-cell column flotation unit that is not equipped with a gas disengagement chamber.

The second problem with a pipe distributor was revealed by CFD simulation and confirmed in physical testing. The fluid

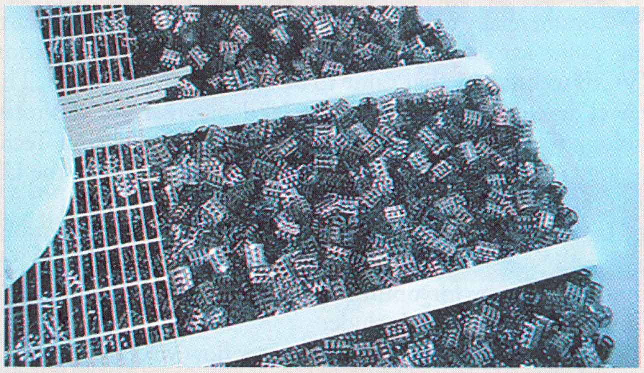

Fig. 3. NATCO-lescer J-VSP coalescing media in a 40,000-bwpd VCF.

velocity in a header and lateral distributor tends to push water from the distribution piping in a direction that is not perpendicular to the distributor pipe, resulting in a non-uniform fluid distribution within the vessel.

To correct both problems, the cyclonic inlet design was adopted for use in the column flotation unit. A proprietary Tulsa University Separation Technology Projects (TU-STP) Consortium GLCC design program was used to size the inlet device for gas slug release while retaining smaller gas bubbles (< 500 microns), for their ability to contribute to the first stage of gas flotation. In addition, the cyclonic inlet introduces the contaminated water to the flotation cell with a swirl pattern. The swirl pattern discourages development of stationary, vertical circulation cells that would permit oily water to by-pass columns of flotation gas bubbles.

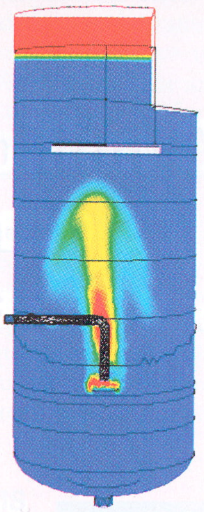

Fig. 4. A CFD simulation shows that a commercially available eductor design would be ineffective at distributing gas within a VCF cell. This was later verified by physical testing at the NATCO-Axsia UK test facility.

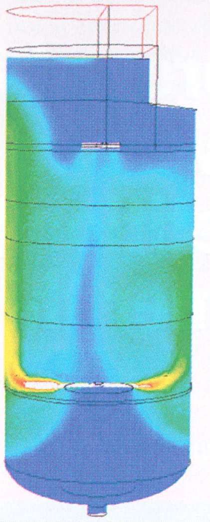

Fig. 5. By using a radial distribution pattern, gas bubbles can be distributed over a large area in a column flotation cell. The area over which gas can be distributed is controlled by the geometry of the radial eductor assembly.

Providing a uniform, reliable distribution of flotation gas bubbles. CFD simulations showed that properly designed hydraulic eductors could provide an excellent bubble pattern for flotation in a column configuration, if the gas bubbles were introduced via a horizontal, radial pattern. By controlling the precise geometry of the eductor, gas and recycled water could be distributed over a diameter of several feet. Conventional eductors used in horizontal flotation cells did not have this radial distributive capability.

Fig. 4 shows a CFD simulation, later confirmed by physical testing, of a commercially available eductor. The simulation shows the rapid rise of poorly distributed gas bubbles in a column that would by-pass most of the oily water in a column flotation cell. Fig. 5 is from a CFD simulation that shows how a radial distribution pattern of water and fine gas bubbles can be designed to disperse gas bubbles widely in a flotation cell. In this case, the gas bubbles are distributed over a diameter that exceeds 5 ft. In Fig. 6 is a photo of a radial eductor that is installed in a 10-ft diameter VCF now in service in the Gulf of Mexico.

Minimizing the size of gas bubbles for flotation. Fig. 7 is an example of the bubble size distribution that was measured near the outlet of a VCF radial eductor prior to any significant gas bubble coalescence in the flotation cell. As bubbles rise and interact, they will grow, both by coalescence and as a result of the reduction of hydraulic pressure.

Testing at the NATCO-Axsia facility using 2% NaCl brine at ambient temperatures indicated the initial bubble size distribution for the radial eductor's gas bubbles resulted in an average bubble diameter of 250 to 350 microns. This diameter is a function of the produced water's physical/chemical characteristics, such as salinity, hardness, surface tension, and temperature (viscosity), and will thus vary from location to location.

The question arises as to how small the gas bubbles should be for effective flotation. Based upon a column flotation design parameter of ≤2.0 ft./min., average downward water flow, a gas bubble diameter greater than about 125 microns is required for the bubbles to rise up against the downward flowing water.

The importance of small gas bubbles for flotation efficiency can be illustrated by calculating a parameter referred to as the Sweep Factor. The Sweep Factor is the number of times per unit of time (e.g., number of times per minute) a given volume of water is swept by a gas bubble. The larger the Sweep Factor, the higher the probability of successful gas bubble and contaminant interaction, and thus the higher the flotation efficiency. The Sweep Factor is defined by the following equation:

$$\text{Sweep Factor (min}^{-1}\text{)} = (A_{gas} \times F_{gas}) / A_{cell}$$

Where A_{cell} = cross-sectional area of flotation cell (sq ft)
F_{gas} = gas flow rate (cu ft/min)
A_{gas} = total cross-sectional area of gas bubbles per unit volume (sq ft/cu ft)

If gas is flowing into an IGF unit at the rate of 1 ACF/bbl of produced water capacity, the Sweep Factor as a function of bubble diameter is as follows:

Bubble diameter (microns)	Sweep Factor (min^{-1})
100	1,634
120	1,362
200	817
300	545
400	408

These numbers clearly indicate the advantage of introducing smaller gas bubbles into a flotation cell. However, fluid flow considerations, as discussed above, serve to limit the minimum size that can be allowed in a flotation cell. In column flotation, the bubble size introduced must be as small as possible while maintaining the required, net, upward rise velocity for the gas bubble. This is achieved with the VCF radial eductor.

Minimizing internal recirculation zones that by-pass oily water around rising gas bubbles. This issue is addressed by the horizontal, swirling flow pattern from the cyclonic inlet, by the presence of the NATCO-lescer coalescing

Fig. 6. Radial eductor installed in a 40,000-bwpd VCF.

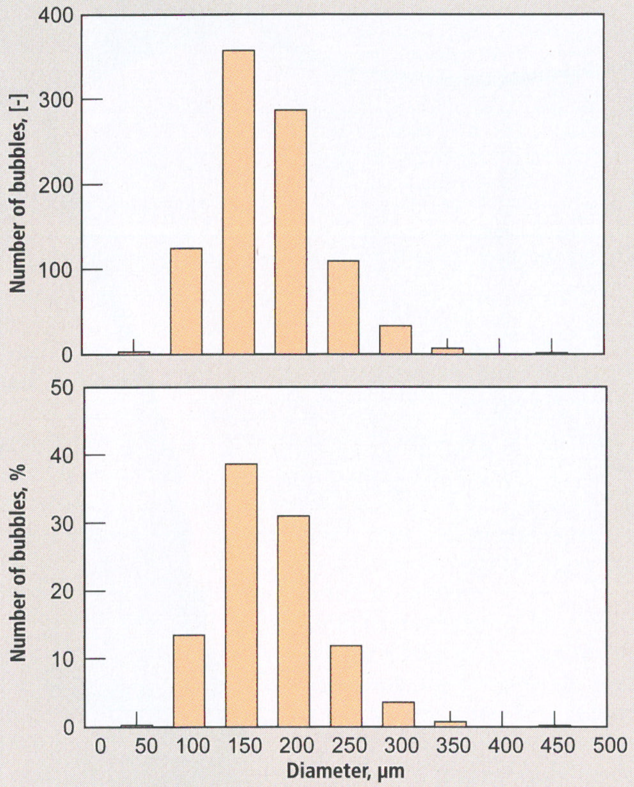

Fig. 7. Gas bubble size distribution from the radial eductor shows a median bubble size near 150 microns.

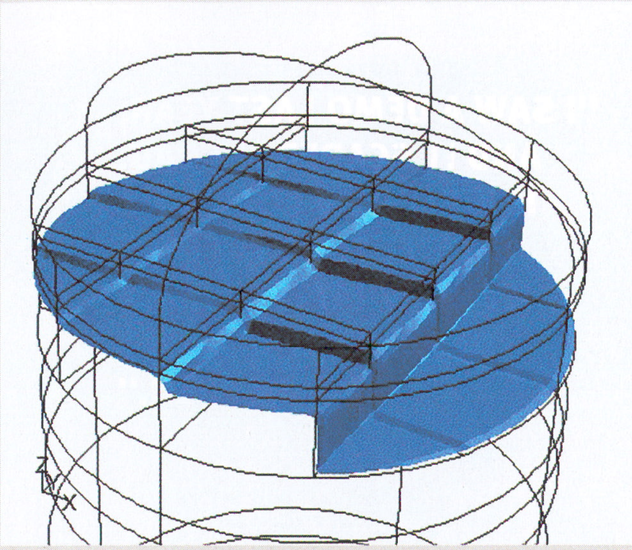

Fig. 8. A two-stage skimming arrangement is used in the VCF unit to control skimming when liquid is sloshing, due to vessel movement along with a floating platform.

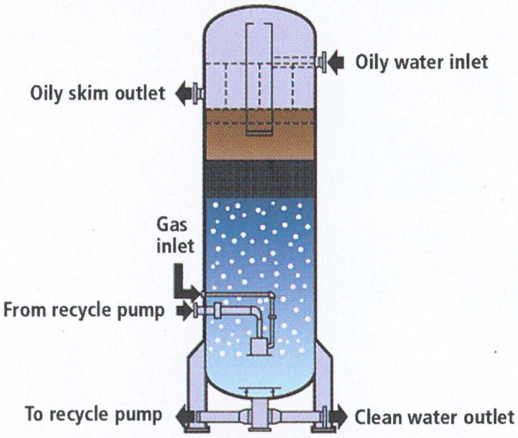

Fig. 9. A schematic diagram of a VersaFlo vertical flotation unit. US and PCT patents pending.

pack, and by the uniform distribution of gas bubbles from the radial eductors.

Control of oil/water skimming in the flotation cell during sloshing induced by movement of the host platform. When installed on a floating platform, an IGF vessel will experience considerable movement. This results in water sloshing within the vessel that can seriously disrupt removal of floating contaminants from the unit. Several CFD simulations using a variety of anti-slosh baffle configurations were studied with little success in the control of the actual fluid sloshing. This was attributed to the fact that, although sloshing can be severe, the actual volume of water that moves to generate this sloshing is small, making it difficult or impossible to control with conventional baffling.

CFD simulations indicated that the most effective method would be to equip the column IGF unit with a two-stage skimming system as illustrated in Fig. 8. In the first stage, water and oil slosh into shallow buckets that effectively dampen water sloshing. Water can leave these buckets via bottom holes, but floating contaminants are retained. The contaminants can then be skimmed into the oil bucket for retention prior to being removed from the vessel.

Elimination of short-circuiting to the outlet nozzle. The design of an outlet nozzle is the final critical VCF component. Without proper design, down-coning of water to the outlet from the center of the cell would induce short-circuiting of fluids through the cell and impede flotation efficiency. To ensure that this does not happen, VCF is equipped with a

Fig. 10. A 10,000-bwpd VCF unit for deepwater Gulf of Mexico installation.

TABLE 1. Field performance data, 10,000-bwpd VCF system.

(The unit is installed on a spar in the Gulf of Mexico. Data were acquired during sea states that varied from 5-ft to 10-ft wave heights.)

Date	Inlet TOG	Outlet TOG
02-03	52 mg/liter	27 mg/liter
02-03	48	19
02-03	37	13
10-04	47	15
10-04	55	23

TOG = Total oil and grease as determined by EPA 1664

capped outlet nozzle that distributes the inlet flow over a large fraction of the cell's cross-sectional area.

FIELD PERFORMANCE

A schematic of 10,000-bwpd, standard VCF design is shown in Fig. 9. This unit, shown photographically in Fig. 10, has been operating in the Gulf of Mexico for over 18 months. Fig. 11 depicts a 40,000-bwpd unit, along with its upstream hydrocyclone package installed on the same skid.

Fig. 11. A 40,000-bwpd VCF flotation cell, along with its upstream hydrocyclones, is prepared for installation on a TLP in the Gulf of Mexico.

Performance data for the 10,000-bwpd column flotation unit, based upon the above-described design principles, are illustrated in Table 1. The unit is installed on a spar and experiences considerable movement due to the long moment arm between the spar's center of rotation and the point at which the IGF unit is installed. Nevertheless, with the proper application of chemistry to assist with contaminant coalescence and flocculation, the quality of the produced water remains well within permissible discharge limits. The larger VCF unit has been operating for a shorter period of time at below designed water flowrates. **WO**

THE AUTHORS

Ted Frankiewicz is VP for Liquid Process Solutions at NATCO. He holds a PhD in physical chemistry from the University of Chicago. Dr. Frankiewicz has 27 years of oilfield-related experience with Occidental Petroleum, Unocal and NATCO. He holds 15 patents and has published more than 30 papers in a variety of technical journals. His expertise is in process systems, separator design, and surface and colloid science.

Chang-Ming Lee is a senior CFD consultant in the Process Solution Group at NATCO's headquarters in Houston. He studied at Texas A&M University and earned an MS degree in chemical engineering and a PhD in interdisciplinary engineering. He has been exclusively responsible for managing CFD projects at NATCO and is a key contributor to the firm's ongoing technological advancements.

Kevin Juniel is product manager, Water Treatment Solutions, at NATCO in Houston, a position he has held since January 2001. Prior to this, he was the technical manager of Axsia Serck Baker Inc. in Houston. Mr. Juniel began his oilfield career with Conoco, where he spent 15 years in both Production Engineering and Process Engineering roles. He holds a BS degree in chemical engineering from the University of Wisconsin, Madison, and has authored numerous papers relating to water treatment.