JA187

Computer Simulation Helps Improve Vertical Column Induced Gas Flotation (IGF) System

Computer simulation has helped NATCO engineers make dramatic improvements in a single-cell vertical induced gas flotation (IGF) system used to clean up oily water that is co-produced with oil and gas. Single cell-vertical column flotation systems represent a major design challenge because of the need to accomplish the same level of separation in a single cell that is achieved in four cells with a conventional horizontal tank design. This type of unit is very difficult to design using conventional physical testing methods because building prototypes is costly, takes a long time to build and it provides very limited information on why the design did or did not work. During the latest design project, NATCO research engineers, led by Dr. Chang-Ming Lee, used computational fluid dynamics (CFD) to simulate the performance of the initial design concepts prior to the prototype stage. Engineers used the latest and most extensive Eulerian multiphase model of FLUENT CFD software to simulate both water and gas bubbles of various sizes within the IGF system. The CFD simulations predicted, and physical testing conducted at NATCO's laboratory in the United Kingdom verified, that bubble distribution patterns from the initial design required considerable improvement. The simulations provided great insight into the characteristics of the fluid flow in the IGF system. These preliminary results were used to modify hardware designs for subsequent physical testing, finally resulting in a finished product, now under the

Figure 1: Photo of a VersaFlo^{TM} unit (capacity of 6,400 m^3/day) ready to be shipped to a deep water platform in The Gulf of Mexico

trade name of VersaFlo™, that has achieved far better performance than previously existing products.

Water is produced along with oil from most wells and typically must be cleaned to a purity of less than 29 parts per million of total oil content prior to disposal. IGF systems work by introducing small gas bubbles from 100 to 500 microns in diameter into the vessel with contaminated water. By controlling the volume and rate of gas and water introduced, the development of the proper bubble sizes for efficient lifting of oil droplets and suspended solids is achieved, explains Dr. Ted Frankiewicz, Vice President, NATCO Process Solutions Group. The hydrophobic oil droplets and oil-coated solids attach to the water-gas interface and float to the surface as the gas bubbles rise. The floating contaminants are then removed by a skimming mechanism.

In the traditional IGF design, oily water enters the unit from one end and passes sequentially through a series of flotation cells. To save weight and space on offshore platforms, the industry is moving towards using a single-cell vertical column configuration. It is much more difficult to obtain the required level of separation in this type of configuration, which typically requires removing 90% of the oil contamination. With a single cell, there is only one chance to remove the oil droplets, and the residence time in a typical unit is limited to four minutes in order to meet volume requirements. It is critical to obtain effective dispersion of the gas bubbles in the vessel so that they contact all of the contaminated water in the vessel.

Problems with the conventional approach

NATCO engineers were determined to improve the performance of existing vertical column floatation systems, which typically suffer from short-circuiting, resulting in poor separation efficiency. Engineers realized it is a difficult challenge in solving these problems because of the drawbacks of the traditional build-and-test method that is normally used for IGF design. Building and testing prototypes is expensive and time-consuming because they should be constructed to full size in order to avoid scaling discrepancy. Another problem is that the information obtained from testing prototypes is often limited. Engineers are typically able to obtain flow and pressure measurements at only a few spots inside the vessel; therefore, it's very easy to miss problems in other locations that might have a negative impact on the overall performance. As a result, each design iteration typically provides very little information that can be adapted to improve the next one.

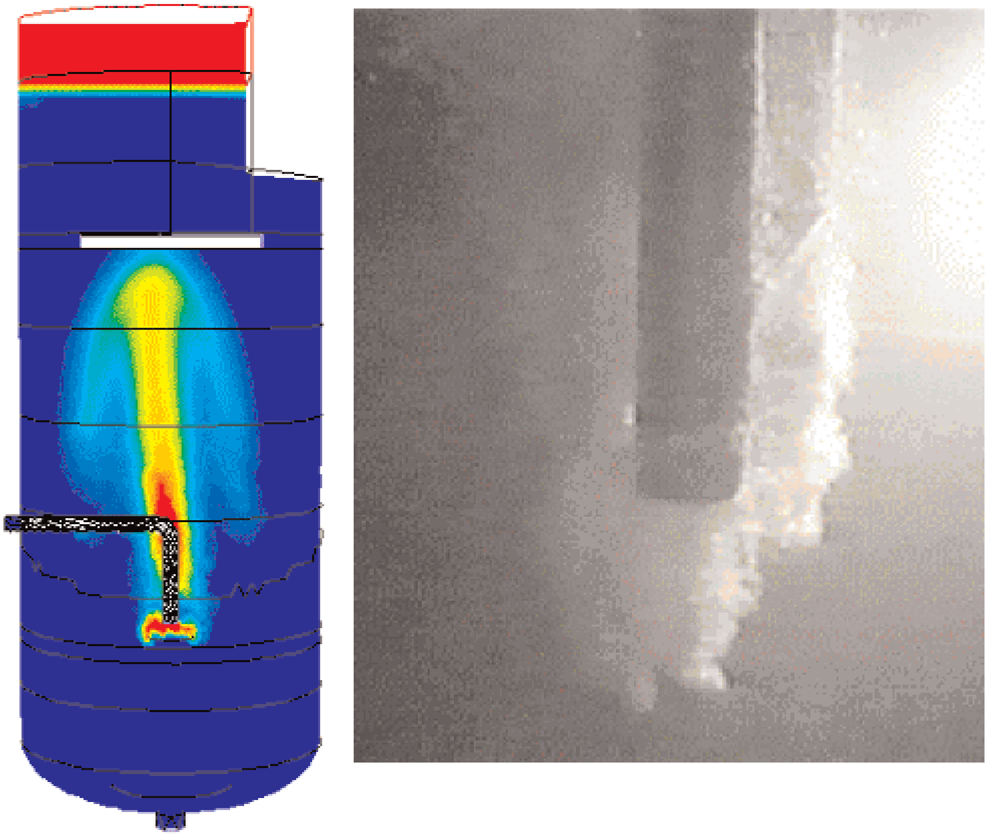

Figure 2: Comparison of CFD simulation and physical laboratory testing

NATCO's engineers took advantage of using CFD to evaluate the performance of a large number of designs prior to the prototype phase. CFD simulation results provide fluid velocity, pressure, particle trajectories, and other relevant variables throughout the entire solution domain for vessel

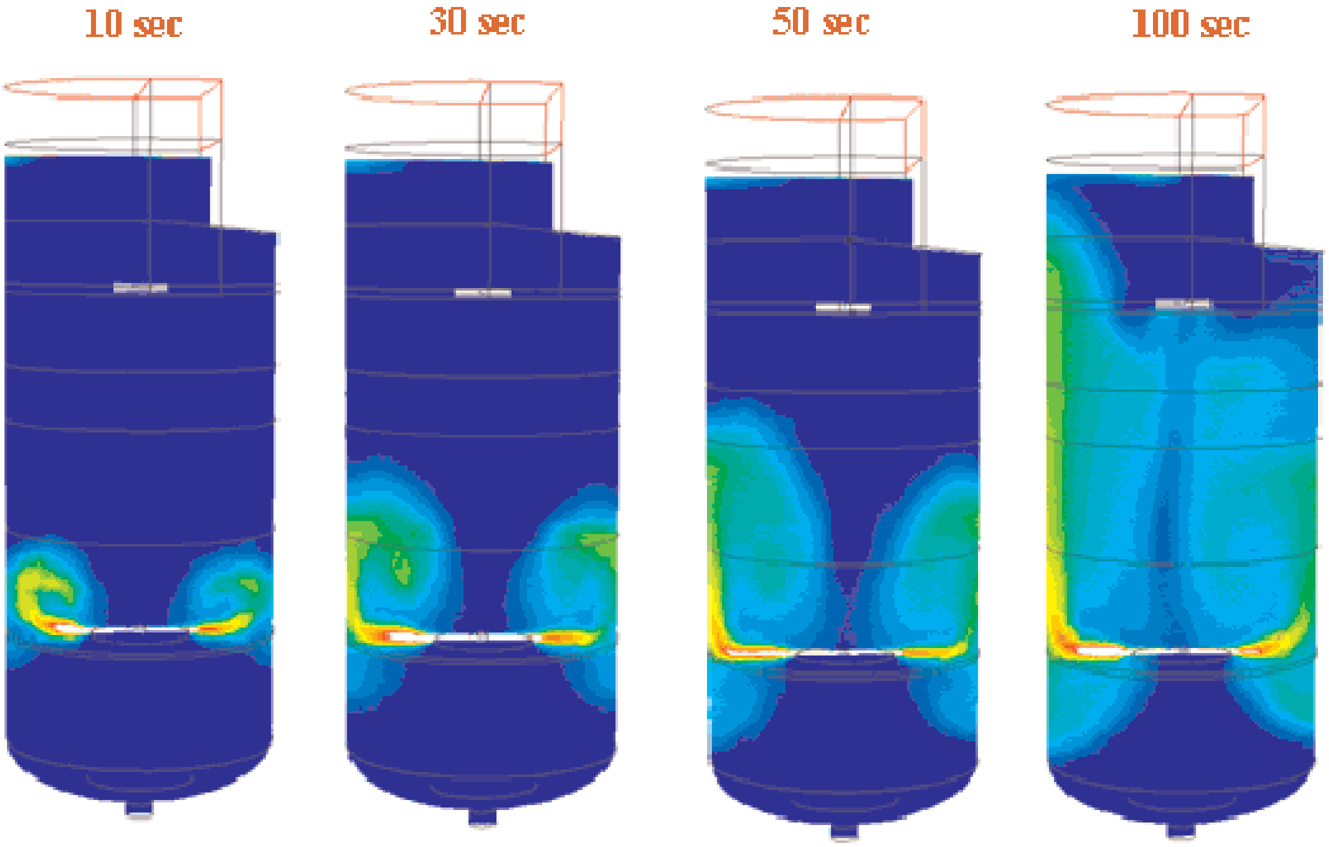

Figure 3: Improved air bubble distribution provided by a new patent pending eductor design

models with complex geometries and boundary conditions. As part of the analysis, a vessel designer may change the system geometry or the boundary conditions and view the distribution of fluid flow patterns or the effects of other variables. NATCO engineers, led by Dr. Chang-Ming Lee, selected FLUENT CFD software from Fluent Inc., Lebanon, New Hampshire, because FLUENT has pioneered the modeling of complex multiphase flows. FLUENT Version 6, which was used in this study, is capable of modeling the hydrodynamics of multiphase mixtures with a wide range of bubble sizes and FLUENT has accurately simulated the distribution of gas bubbles as they move through the bulk liquid phase.

Improving the gas bubble injection device

The initial models focused on defining the configuration of the eductor, a gas bubble injection device, and the size of gas bubbles it introduced into the IGF vessel. Eductors are designed to efficiently generate gas bubbles by discharging fluid via a Venturi tube, creating a partial vacuum that draws gas into the eductor. The initial simulation results were surprising to everyone involved in this study because they showed that the distribution of bubbles in the IGF vessel was much poorer than had been expected when using the commercially available eductor. Rather than covering the cross-section of the vessel, as is required to avoid short-circuiting, the bubbles released from each eductor mostly remained within a fairly small cylindrical area, creating a straight fast-rising plume as they travel up to the top of the vessel. Just as discouraging, the uneven distribution of bubbles across the tank created recirculation zones, which further increased the tendency towards short-circuiting. This helped to explain why the performance of many earlier designs from competitor companies had been disappointing. Because this was the first time that the design team had simulated this type of device, physical tests were performed to validate the CFD model and to verify its accuracy. Figure 2 illustrates the comparison between CFD's prediction and laboratory testing.

NATCO engineers then began a series of about forty additional CFD design variations with the goal of improving the separation efficiency of the IGF design concept. Several simplifying assumptions were made in the model. The 3-D geometry was based on a test vessel located at their laboratory in Gloucester, UK, which is approximately 60 inches in diameter and 12 feet in height. The oily water inlet and air/water mixture inlet were simplified as boundary conditions representing the real positions inside the vessel, hence eliminating the need to model the inlet piping as well. Internal supports and the coalescing packing material were also ignored. FLUENT's Eulerian multiphase model with the unsteady segregated solver was used to simulate the mixture flow of air and water. The volume of fluid model with the geometric reconstruction scheme was later used for a separate wave motion suppression study. To improve the spreading of bubbles into a uniform cloud that would contact most of the cross-sectional-area of the IGF, a number of solution variables were altered in the simulations, including the geometry of the oily water inlet device, the volume of water flowing into the vessel through the eductor, the gas volume fraction in the eductor flow, the size of the gas bubbles, and the internal baffle configurations of the vessel.

Iterating to a superior design

In the first stage of the study, NATCO engineers performed steady-state single phase test runs involving only oily water flowing downward through the tank in order to be certain that the downward oily water flow stream can be adequately controlled within the desired range. In the next stage, the eductor flow into the IGF vessel was modeled with a high air volume fraction while varying the liquid levels and fluid flow rates. This series of simulations demonstrated flow patterns that matched up well with physical experiments, helping to provide further confidence in the accuracy of the simulation. NATCO engineers then modeled different configurations of the gas inlet devices while continuing to vary the gas volume fraction. Through this process, design engineers were able to gain considerable understanding of how bubble distribution over the cross-section varies with different inlet design configurations. As a result, NATCO engineers were able to develop a new patent pending eductor design that offers performance far superior to conventional designs used in the IGF system. The new eductor design features a more radially-directed fluid discharge as opposed to the strictly downward discharge found in conventional designs. Figure 3 shows how the new gas inlet configuration promotes uniform bubble distribution across the IGF test vessel.

Subsequently, NATCO engineers experimented with different configurations of internal baffles in an effort to obtain additional performance improvements. The internal baffle plate is designed to control the downward flow of the oily water and to prevent the recirculation zones. It was observed that by restricting the area in which the oily water can flow downward, the baffle promotes a more uniform gas bubble distribution. NATCO engineers went one step further by including user-defined functions to simulate the six degrees-of-freedom ocean wave motion induced sloshing effect, and how it will affect the performance of the IGF vessel to be installed on a floating offshore platform. Improved wave motion suppression baffles were later carefully installed in the skimming region inside the offshore version of IGF system. Table 1 lists the actual field performance data from a VersaFlo™ 10,000 BBL/Day offshore IGF system.

Field Data (mg/L)	Inlet TOC*+ (mg/L)	Outlet TOC (mg/L)
Test #1	52	27
Test #2	48	19
Test #3	37	13
Test #4	35	13

Table 1. Offshore IGF System Field Performance Data

***The sea state was at about 8 - 10 ft.**

+TOC: Total Oil Content.

The final system configuration, now with US and PCT patents pending, dramatically improves upon existing designs by providing a uniform distribution of gas bubbles across the vessel while effectively eliminating the recirculation zones. These improvements were achieved because the quickness of the CFD simulation made it possible for design engineers to examine far more design variations than would have been possible using the conventional build-and-test method, and also because CFD simulation provided more insight than physical testing could on the performance of each design. CFD is now playing a critical role to NATCO's new product design.

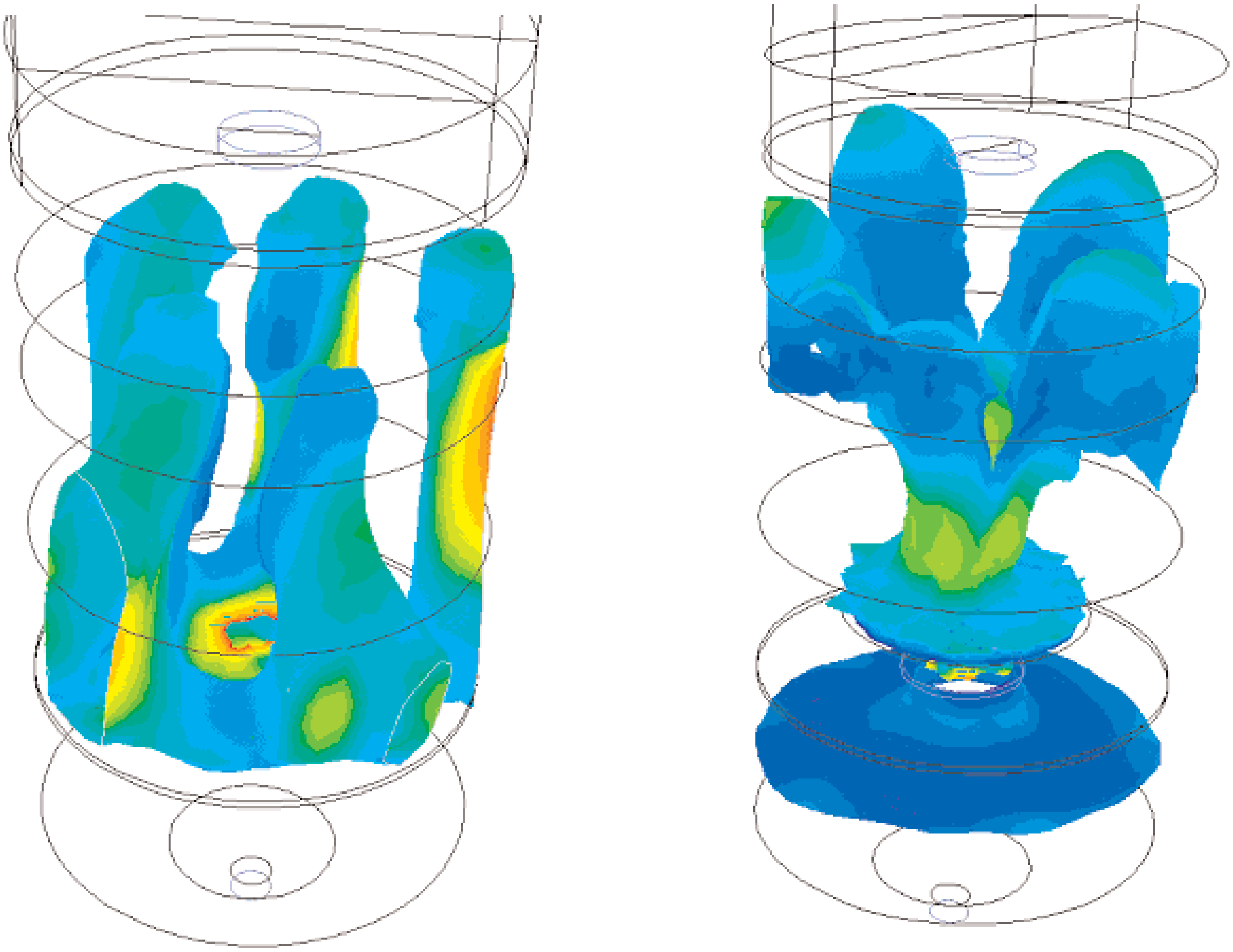

Figure 4: The case without the baffle (left) and with the baffle (right); The internal baffle plate helps eliminate recirculation zones