Using Computational Fluid Dynamics (CFD) Simulation to Model Fluid Motion in Process Vessels on Fixed and Floating Platforms

Dr. Ted Frankiewicz
Dr. Chang-Ming Lee

NATCO Group
Houston, TX USA

IBC 9th Annual Production Separation Systems Conference
London, U.K.
June 2002

ABSTRACT

Computational Fluid Dynamics (CFD) is a mathematical tool capable of simulating a wide range of fluid flows. Integrated CFD software has been applied to study the flows in oilfield separators. The influence of inlets, internals, and outlets has been studied with CFD simulations. Of particular value has been the ability of CFD to simulate wave induced sloshing in vessels mounted on floating platforms. The design and placement of baffles to mitigate liquid sloshing was determined by using CFD to simulate fluid flows in such vessels. The simulations accounted for the movement of the vessel based upon its location on the platform and included the influence of fluid flows on slosh motion

INTRODUCTION

Until recently, the engineering design of two- and three-phase separators was considered to be mature. However, the need for separation systems to operate in challenging environments, such as on floating platforms, has led to a demand for improved vessel designs that require reduced fluid residence times and effective separation even when the fluids are jostled by the six-degrees of motion (pitch, roll, heave, surge, yaw and sway) experienced by a floating platform. CFD allows a designer to observe the simulated fluid flow paths within a vessel and assess how the separator will perform under these difficult circumstances.

The application of CFD to solving oilfield related problems can be of value in new product development, vessel design optimization, determining sources of underperformance for existing or proposed vessels, and for evaluating the projected performance improvement when retrofits are installed to upgrade existing process equipment [1]. CFD provides the means to visualize fluid flows within a separator as well as an ability to track the movement of gas/liquid and liquid/liquid interfaces. In newer versions of CFD software, gas bubbles, oil droplets, and solid particles can be tracked through a separator using advanced multiphase models. This permits one to incorporate performance enhancement into the design a vessel without the need for extensive testing in physical models. Validation of simulated results is recommended when possible, however, to insure that the CFD simulations are in fact providing reasonable predictions.

CFD SIMULATION PARAMETERS

The simulations reported here were based upon standard "k-ε" turbulence and volume of fluid (VOF) models.[2] Both steady state and transient simulations were used depending upon the objective of a particular project. The CFD software package used was from Fluent Incorporated (Lebanon, New Hampshire). GAMBIT was used to build the model geometries and their volume meshing. FLUENT was used to run the simulations, and FLUENT's parallel processing capabilities were implemented to accelerate the simulation process.

To model the movement of the vessels on floating platforms, a special User Defined Function (UDF) was developed. The UDF allowed the movement of the separator of interest to be calculated based upon its actual location on the floating platform, the Center of Rotation (COR) of the platform, and the periods/amplitudes for each sea-state induced motion. Typically, the sea-state data for the simulation included 1-year, 10-year and 100-year storm conditions.

Parameters that were derived from the simulation included "drag coefficients"[3] at specified wall surfaces within a vessel, average and/or maximum pressure on the surface of an internal component, and fluid velocity profiles. Although these parameters were useful for comparing the intensity of motion on a relative basis, the most instructive information was generally derived from the 2D and 3D animations that showed simulated fluid movement in response to a vessel's motion. The animations were augmented with particle tracking, a study of fluid velocity vectors and the distribution of turbulence intensity.

RESULTS AND DISCUSSION

The key components in a typical separator that control fluid path line development include the inlet nozzle, the inlet momentum-breaking device, perforated plates, weir or bucket faces, and outlet nozzles. Some components have been modeled individually as well as in combination during the course of the CFD studies.

Perforated plates are used in separators both to establish good fluid flow distribution, and to control liquid sloshing. In a CFD simulation, the perforated plate is modeled as porous media of finite thickness with directional permeability. The fraction of open area can be varied in the porous zone, but hole sizes in the plate are not specifically modeled. To validate the assumption that flow through porous media approximates that through a perforated plate, a separate CFD study was conducted on the design and engineering of perforated plates.

To illustrate how a perforated plate impacts fluid flow in a vessel, Figures 1A and 1B show the results from a 2-D study where in the inlet liquid hits a splash plate and is directed downward. In Figure 1A, the fluid flow path lines tend to be confined along the bottom of the vessel until fluid begins to approach the oil bucket. As can be seen, the flow path bypasses a significant portion of the separator's liquid volume, shortening effective fluid residence time. A general rule of thumb that has emerged from CFD studies is that the fluid at the inlet tends to anticipate the outlet and takes the path of least resistance – often a path not always obvious to a design engineer.

Figure 1B shows how the installation of perforated plates impacts the distribution of fluid flow. The first plate assists the development of uniform flow across the entire liquid cross section. The second plate blinds the fluid, as it flows through the middle of the vessel, from the outlet and effectively eliminates the tendency for short-circuiting.

Figure 2 shows path lines calculated in a 3-dimensional CFD simulation for fluid flows in a laboratory scaled chamber. In this study, physical testing was used to validate the CFD simulation results. The liquid enters the chamber through an off-center inlet. The perforated plate redistributes the flow, but flow path lines develop quickly downstream of the perforated plate and fluid short-circuiting is both predicted by the simulation and observed in laboratory tests.

The type of floating platform on which a separator is installed has a significant impact on the sloshing of fluid inside the separator. Figures 3A and 3B show representative locations for a large separator on a TLP and on an FPSO as well as the Center of Rotation (COR) for each platform. The length of the moment arm, i.e., the distance from the COR to the separator, affects the intensity of sloshing in a vessel. A separator installed on an FPSO will generally experience more roll motion than that on a spar or a TLP and this roll motion will not always remain in phase with the pitch and surge.

One type of slosh motion control baffle that has been installed in separators on floating platforms is a horizontal ring baffle. According to published research [4, 5], the ring baffle is capable of dampening liquid slosh motion when properly sized and positioned. CFD simulations confirm a minor reduction in slosh amplitude in the presence of a ring baffle, but the reduction is far less than what is required in a typical 3-phase separator. Laboratory validation studies [6] have shown that these horizontal ring baffles can, if improperly placed, generate significant interface turbulence within a vessel that is counter productive for oil-water separation.

Criteria that must be considered for the design and installation of perforated plates include the fraction of opening area of the plate, the size and layout of holes that provide the opening area, the amount of open area under the plate to allow for sand migration, the locations for baffle placement, and the number of baffles required to control flow distribution and/or liquid sloshing. Where possible, the results of these studies were compared with standard engineering calculations [7] to validate the CFD simulations. The fractional open area is a compromise between the need to restrict flow through the plate and the need to minimize oil or water droplet shearing to avoid creation of emulsions.

The complexity of flow through a perforated plate is illustrated in Figure 4. Figure 4 shows the development and decay of fluid jets through holes in a perforated plate. Note that when flow approaches the plate with a strongly non-uniform velocity distribution, the flow downstream of the plate tends to spread. Also, velocity vectors indicate that significant turbulence develops immediately upstream of the perforated plate with substantial recirculation that is dependent upon the size of the holes and the fraction of open area in the plate.

Figure 5 illustrates the high velocity for liquids that can occur under a perforated plate when open area is retained to allow for sand migration. CFD simulation of vessels on floating platforms indicated such flow paths not only favored short-circuiting for water, but it also increase the chance of oil/water mixing if the interface level is too close to the lower opening area.

Figure 6 shows anti-slosh baffle placements within one particular 10 feet diameter by 40 feet (Seam-to-Seam) three-phase separator. Fluid enters the vessel through a Porta-Test Revolution® Inlet Device. Gas-liquid separation takes place within the inlet device and liquid exits through a horizontal circular slot at the lower end of the capped tube, see Figure 7. The tubes are installed in pairs with 6 tubes having been selected for installation in the subject separator.

For the CFD simulation of the separator, the Porta-Test Revolution® inlet devices are simplified as two rectangular blocks with flow entering the vessel from the bottom of the blocks. The first perforated plate is installed just downstream of the Revolution® Tubes in order to redistribute fluids that emerge from the liquid exit slots. The remaining perforated plates are then positioned for control of fluid sloshing.

Figures 8A and 8B show how fluid sloshing is predicted in this separator in response to platform movement during a 10-year winter storm condition. The simulation was performed for the vessel with and without perforated plate baffles. Note that without the plates, the fluid motion within the separator borders on violent (this is more evident when watching the animated sequence of fluid motion that was generated from post processing of the CFD simulation results). However, with slosh suppression plates installed, the liquid motion and interface variation within the separator are dampened considerably.

Finally, the design and location of outlet nozzles also affect fluid path lines in a vessel and, as a consequence, the quality of the discharged liquid. Figure 9 shows how fluid approaches a water outlet nozzle in an oil-over-weir separator. Note that water flowing toward the nozzle comes at least in part from near the oil-water interface with the down-coning actually causing a depression of the oil-water interface in the vicinity of the water outlet. This down-coning drags partially separated oil droplets from near the oil-water interface into the discharge liquid, resulting in a degradation of the quality of the water leaving the vessel.

SUMMARY & CONCLUSIONS

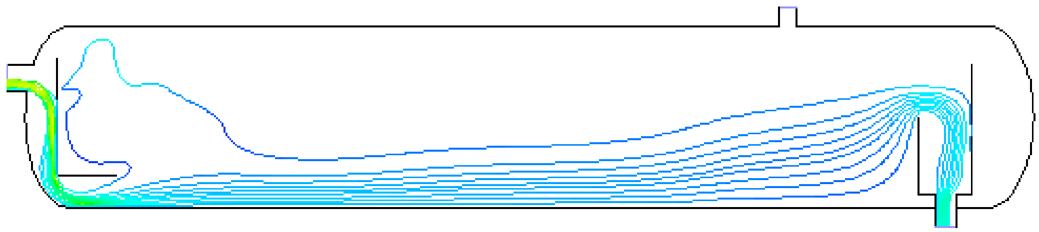
CFD is a powerful mathematical tool for simulating the flow within oilfield processing equipment. CFD simulations allow a designer to visualize how all components of a separator, from the inlet nozzle to the outlet nozzle, affect separation and flows. Using CFD, designs for internals such as perforated plates can be developed and their locations within a vessel can be optimized to control flow distribution and also minimize liquid sloshing for offshore applications.

REFERENCES

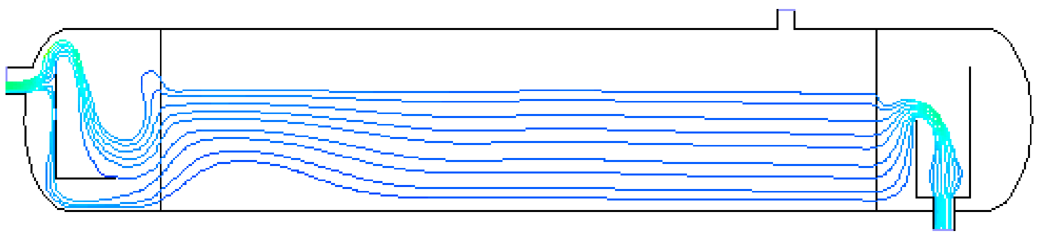
1. Frankiewicz, T. C., Browne, M. M., and Lee, C-M., "Reducing Separation Train Sizes and Increasing Capacity by Application of Emerging Technologies", Offshore Technology Conference Paper 13215, 2001.
2. D. L. Youngs, "Time-Dependent Multi-Material Flow with Large Fluid Distortion", K. W. Morton and M. J. Baines, editors, *Numerical Methods for Fluid Dynamics*. Academic Press, 1982.
3. Fluent 6.0 User's Guide, Fluent Inc., Lebanon, NH, November 2001.
4. Silverman, S., and Abramson, H. N., editors, "The Dynamic Behavior of Liquids in Moving Containers, Chapter 2 – Lateral Sloshing in Moving Containers", Report NASA SP-106, Washington D.C., 1966
6. Silverman, S. and Abramson, H. N., editors, "The Dynamic Behavior of Liquids in Moving Containers, Chapter 4 – Damping of Liquid Motions and Lateral Sloshing", Report NASA SP-106, Washington D.C., 1966.
7. Wallace, H. G., NATCO Laboratory Test Report, "Box-Spar Storage Cell Motion Simulation Study", February 2001.
8. Perry, R. H., Green, D. W., and Maloney, J. O., Editors, *Perry's Handbook Chemical Engineers Handbook, 6th Edition*, McGraw-Hill Book Co., New York, 1984.

ACKNOWLEDGMENTS

The authors would like to thank NATCO Group management in general and Mr. Robert Curcio in particular for supporting this work and for permission to make this presentation. The authors would also like to thank Mr. Gary Sams and Mr. Harry Wallace for their constructive discussions relative to interpreting and validating CFD simulation results and for providing insights that proved valuable in guiding the development of the NATCO's CFD capabilities.



(a)



(b)

Figure 1. A splash plate can direct liquid flow to the bottom of a vessel. Dual perforated plates redistribute the path lines down the length of the separator

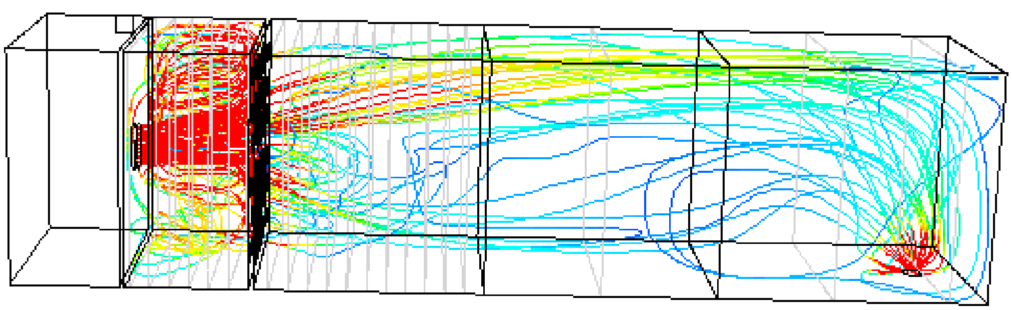


Figure 2. Laboratory model tests validate the CFD calculation of fluid flow path lines, showing how short-circuiting of fluid flow can occur when a single perforated plate is installed in a separator.

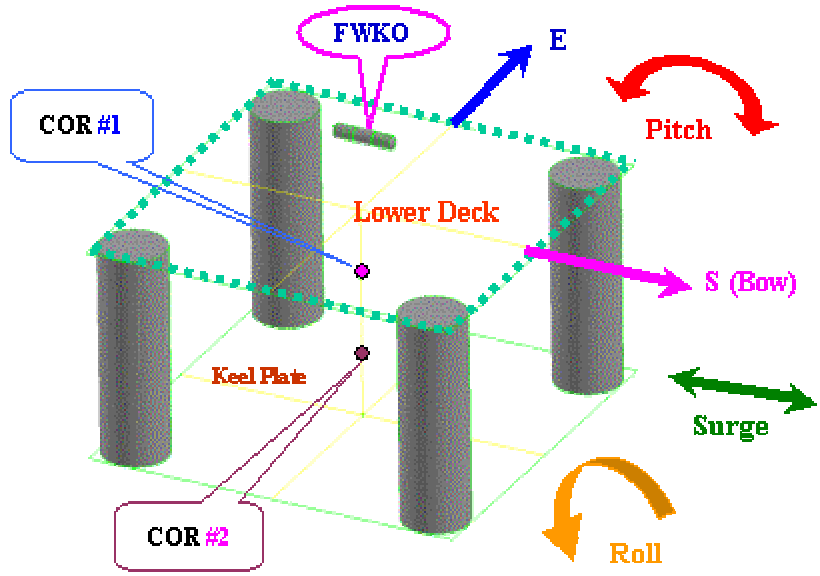


Figure 3A. The placement of a vessel on a TLP is illustrated. Note the position relative to the center of rotation (COR) of the platform.

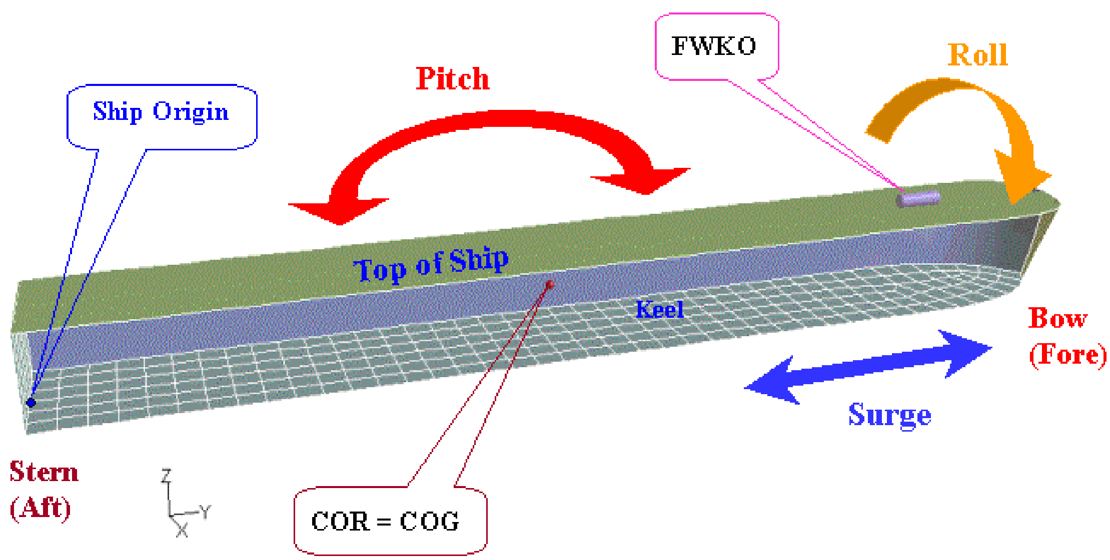


Figure 3B. The placement of a vessel on a FPSO is illustrated. Note the position relative to the center of rotation (COR) of the ship.

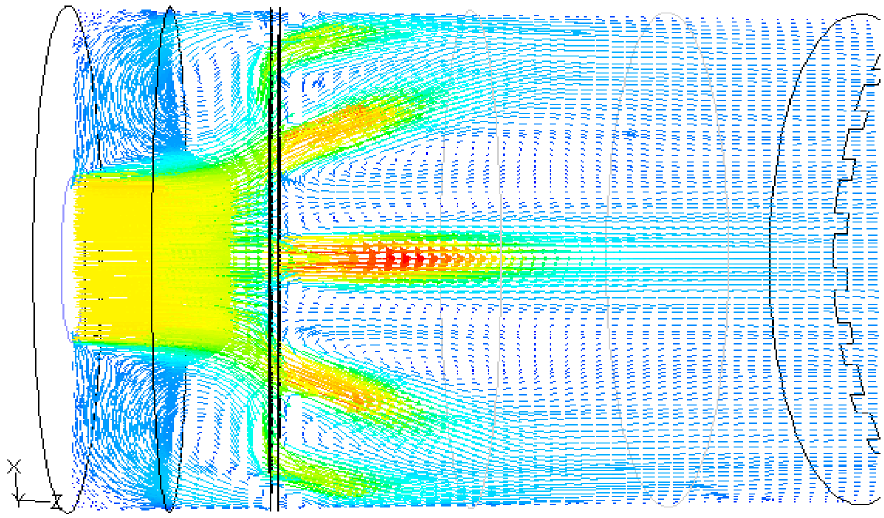


Figure 4. Fluid flow path lines through a perforated plate are illustrated. Note the turbulence and recirculation path lines upstream of the plate.

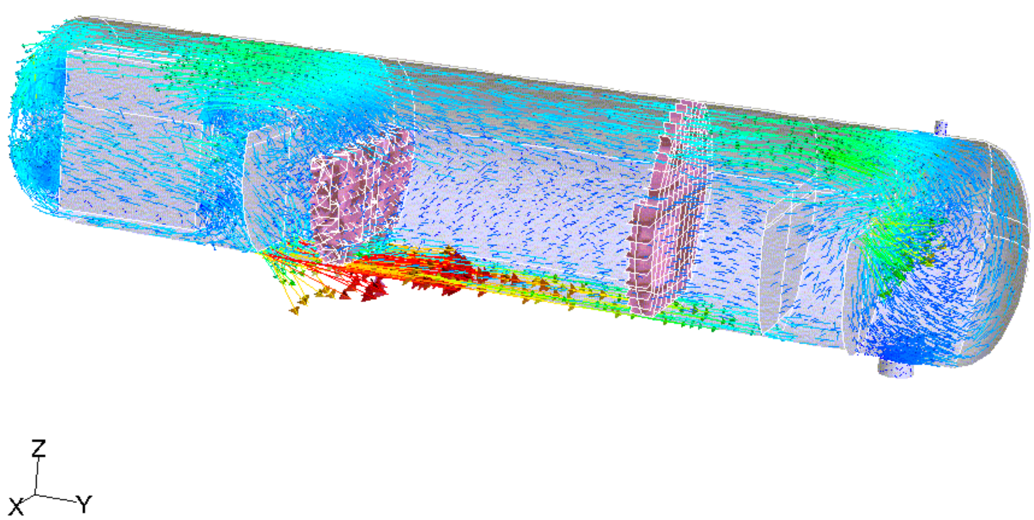


Figure 5. High velocity for water flow can occur under a perforated plate, resulting in reduced water residence time within a vessel.

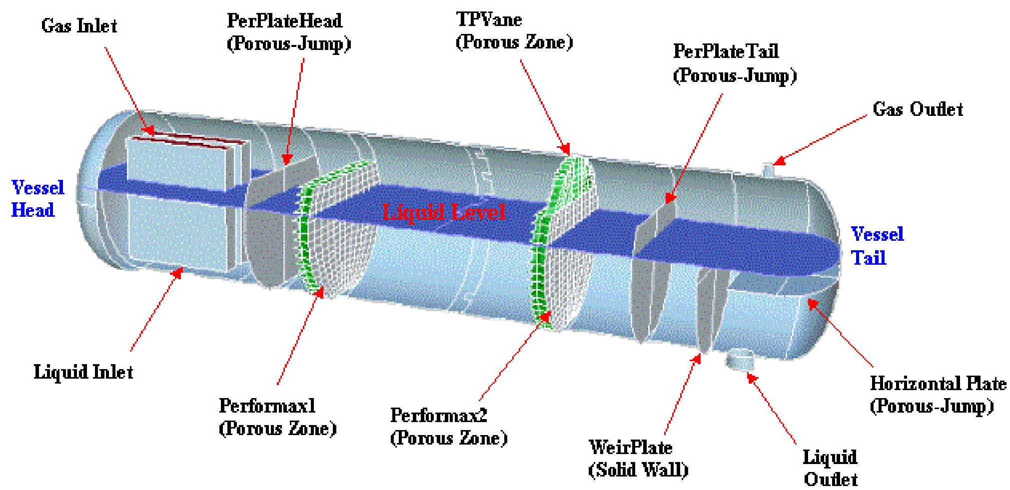


Figure 6. Anti-slosh baffle placements are illustrated for one particular vessel. Baffle location are separator and platform-location specific.

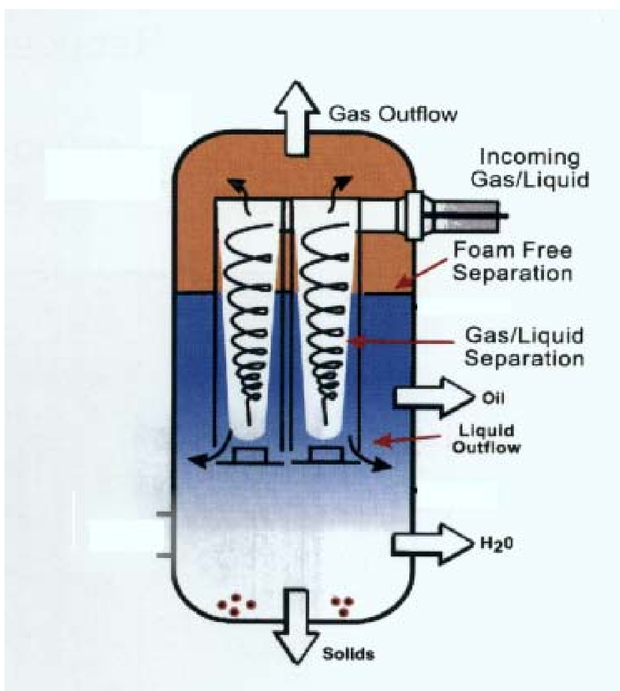


Figure 7. The flow through and separation of liquid and gas in a Porta-Test Revolution cyclonic inlet device are illustrated.

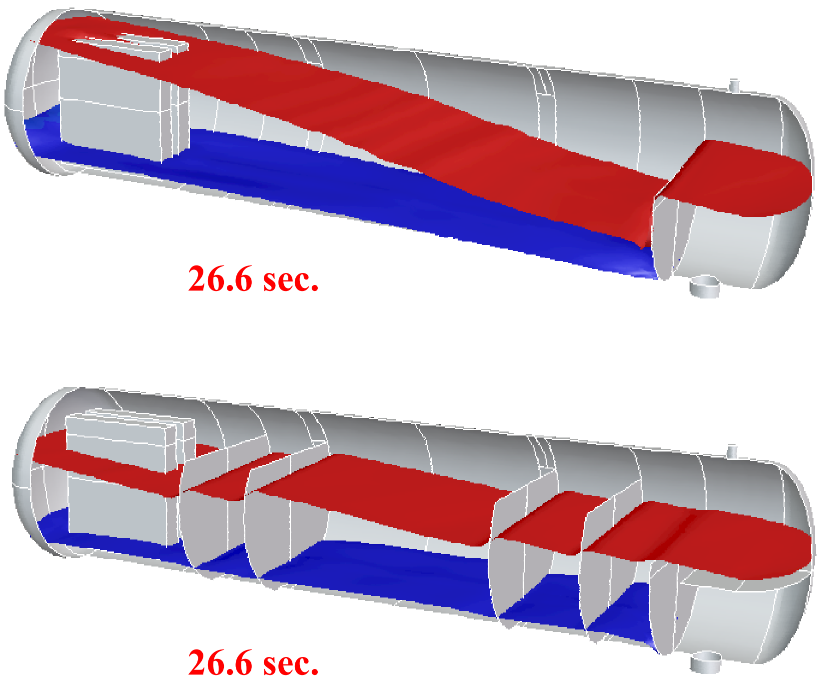


Figure 8. Fluid sloshing in a FWKO on a TLP is shown for the vessel with and without anti-slosh baffling. The drag coefficient on the vessel head was reduced to 38% of its original value by the installation of the baffles, see Figure 9. In both cases, interface positions are shown at simulation time of 26.6 seconds.

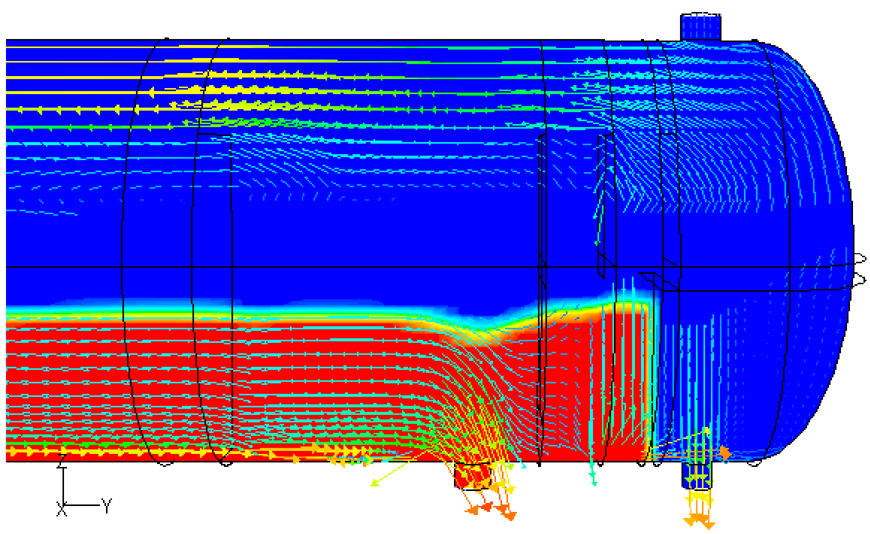


Figure 9. Velocity vectors for fluid approaching a water outlet nozzle illustrate the down-coning of liquid to the nozzle, resulting in reduced discharge water quality.