Use of CFD in Ocean Engineering

Dexin Zhan[1,*], Worakanok Thanyamanta[1] and David Molyneux[1]

[1]*Oceanic Consulting Corporation, A1B 2X5, St. John's, Canada*
**Corresponding author's e-mail:* zhan.dexin@oceaniccorp.com

Abstract

Transportation of oil and gas and oil spills are two potential risks posed to the marine environment. This paper mainly focuses on the use of Computational Fluid Dynamics (CFD) at Oceanic Consulting Corporation (OCC). With OCC's extensive CFD experience in Marine Engineering, it is expected that we could expand our expertise into the area of oil spill forecast and response or other related areas.

Under support of the Atlantic Innovation Fund (AIF), Oceanic has been engaged in a project "Numerical Simulation of Hydrodynamic Loads for the Safe Production and Transportation of Oil and Gas in Harsh Marine Environments" since 2008. This paper presents some CFD studies taken from this project, which includes:

1) 3-dimension flow simulation;
2) Sloshing prediction of a U-tube tank in a vessel; and
3) Moored structure motion in waves and visualization.

It also includes an introduction to some commercial CFD and in-house codes used in Oceanic.

Keywords: CFD, flow pattern, multi-phase, Oceanic, visualization

Introduction

Oceanic Consulting Corporation (OCC), established in 1993, is a commercial marine research and development company. It has focused on the study of ships, boats, offshore structure, or other marine systems using physical modeling and numerical simulation for more than 20 years. Oceanic has become a worldwide leader by delivering a broad and sophisticated range of engineering and consulting service in ocean and Arctic engineering research by collaborating with the National Research Council of Canada Ocean, Coastal and River Engineering, and Memorial University's Ocean Engineering Research Centre and Marine Institute.

The ocean engineering industry in Newfoundland has boomed since oil first start producing from the Hibernia field on the Grand Banks in 1997. There are now three producing oil fields and one more in under development. The environment on the Grand Banks is characterized by strong winds, ice sea states, long periods of poor visibility due to fog and the presence of sea ice. Oceanic wished to be well-positioned to service this industry.

Since 2008, under support of the Atlantic Innovation Fund (AIF), Oceanic has been engaged in a project "Numerical Simulation of Hydrodynamic Loads for the Safe Production and Transportation of Oil and Gas in Harsh Marine Environments". This project focused on numerical prediction of ship maneuvering, seakeeping and ice-structure interaction in an environment of high waves, strong currents, strong wind and pack/level ice. The objective of the project was to develop and validate computer programs that can be used to predict the performance of major components of offshore oil and gas exploration and production systems in harsh environments. The core to the project was Oceanic's in-house codes for ship maneuvering (SML) (Gong, 1993a, 1993b), forces and motions on ships and offshore structures due to waves (MOTSIM) (Pawlowski, et al. 1991) and ice-structure interaction (DECICE) (INTERA 1986).

These in-house codes could be supported by some others that were commercially available and that could also be used for support and validation. For validation purposes, industry acceptance of the code was a key requirement.

In selecting the commercial codes, we were looking for capability to model ships and offshore structures in harsh environments. Particular features required were free surface waves, viscous shear forces and non-linear motion responses, although some linear codes were used for benchmarking studies. Some commercial software, such as STAR-CCM+ (http://www.cd-adapco.com/), FLOW-3D (http://www.flow3d.com/), Hydrostar (Hydrostar 2011), OrcaFlex (http://www.orcina.com/SoftwareProducts/OrcaFlex/), and ShipMo (Gaillarde 2010) have been used and also validated in order to simulate structure motion in the sea states. The aim of this paper is to introduce some progress and achievement related to CFD in the project. It is hoped that the areas of research can be expanded into the area of oil spill forecast and response or other related areas.

CFD Study

1) 3-Dimension flow simulation

Numerical simulation of airflow or water flow around a ship or offshore structure is a basic requirement in marine fluid dynamics. Forces and flow patterns are critical to system performance, and CFD can save money and time when compared to model experiments. A powerful commercial software to model flow is STAR-CCM+.

STAR-CCM+, developed by CD-adapco (http://www.cd-adapco.com/), is an all-in-one CFD solver. It has capability to tackle problems involving multi-physics and complex geometries in ocean and offshore engineering. Some examples of where we have used STAR-CCM+ are to simulate water flow around different vessels (FPSO, high speed vessel etc.), airflow over vessel topside, and flow passing through a combined propeller and rudder unit.

Figure 1 shows a simulation result of flow around a high speed vessel which includes a prediction of the free surface. Figure 2 shows the mesh used for its computation domain. The free surface capture method used in the simulation was volume of fluid (VOF) method. This method considered the fluid in two phases (air and water) to track and locate an air-water interface using a volume fraction which must lie between 0 and 1 in the boundary of air and water.

Figure 3 shows velocity distribution results of flow across an articulated tug barge in three sections and Figure 4 shows oblique flow cross Series 60 vessel in two sections using STAR-CCM+ software.

Figure 5 shows geometry of propeller, nozzle and rudder. Figure 6 shows the simulation of flow interaction between a combined propeller-nozzle-rudder. Figure 7 and Figure 8 show airflow passing over a vessel and its mesh, respectively. This simulation also considered disposition of the exhaust plumes. These two examples show the powerful capability of STAR-CCM+ for a complex geometry.

In the study of oil spill forecasting and response modeling the flow of current and wind is a basic requirement. The STAR-CCM+ is a candidate to predict this type of flow pattern. Furthermore, STAR-CCM+ also has strong ability to treat multi-phase flow simulation.

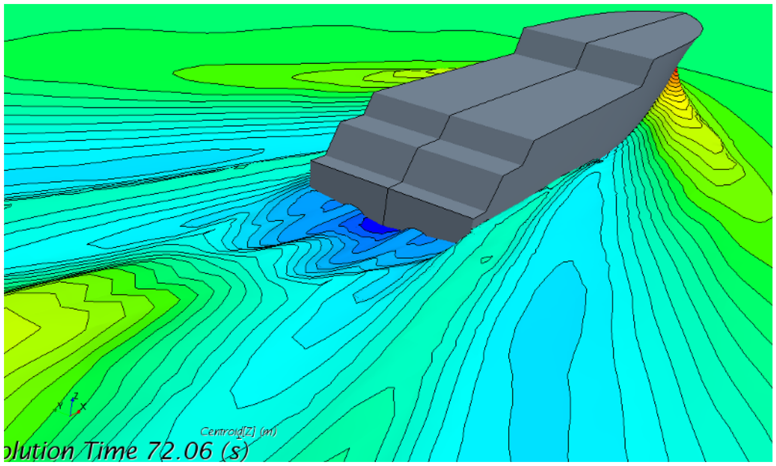


Figure 1. Flow study of high speed vessel

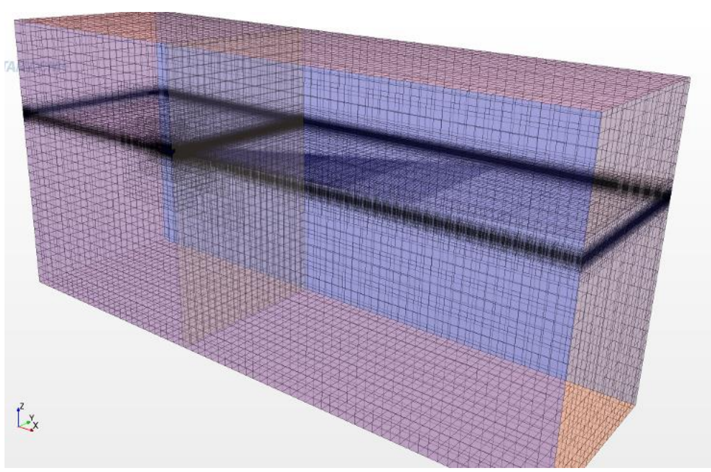


Figure 2. Mesh for high speed vessel, showing fine grid at water air interface

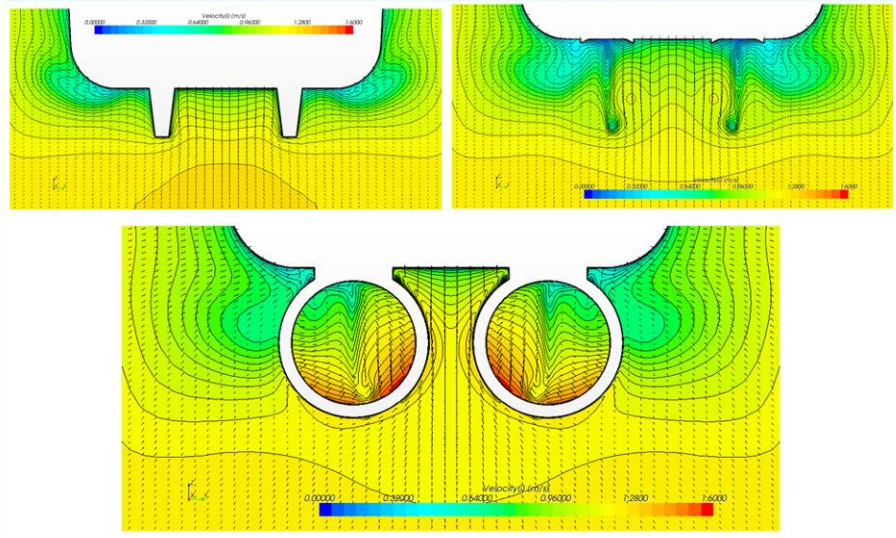
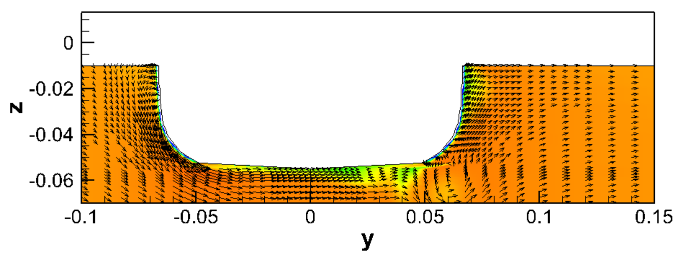
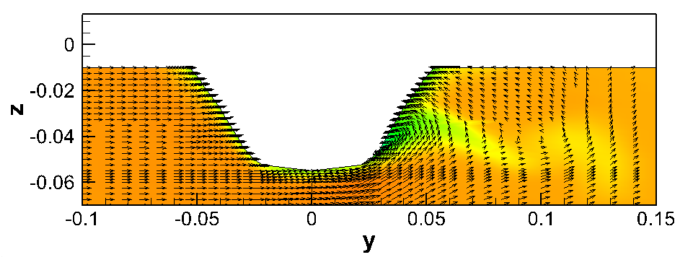


Figure 3. Flow study of articulated tug barge (ATB) – cross sections



(a) Velocity profile at 0.4L cross plane



(b) Velocity profile at 0.8L cross plane

Figure 4. Velocity profiles of **v**essel Series 60 in oblique flow

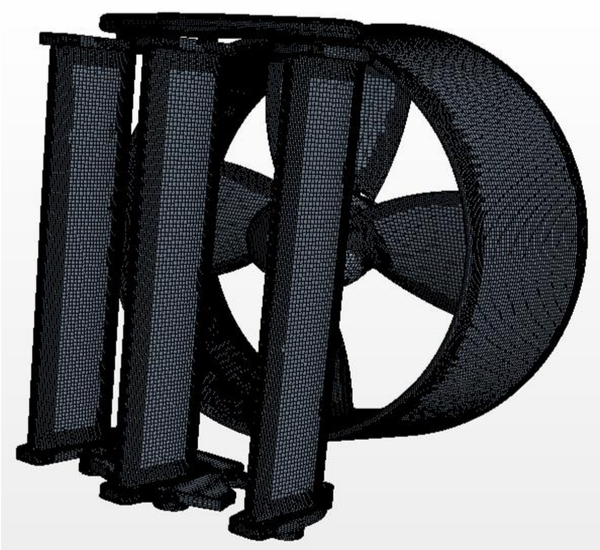


Figure 5. Study of propeller-nozzle-rudder interaction

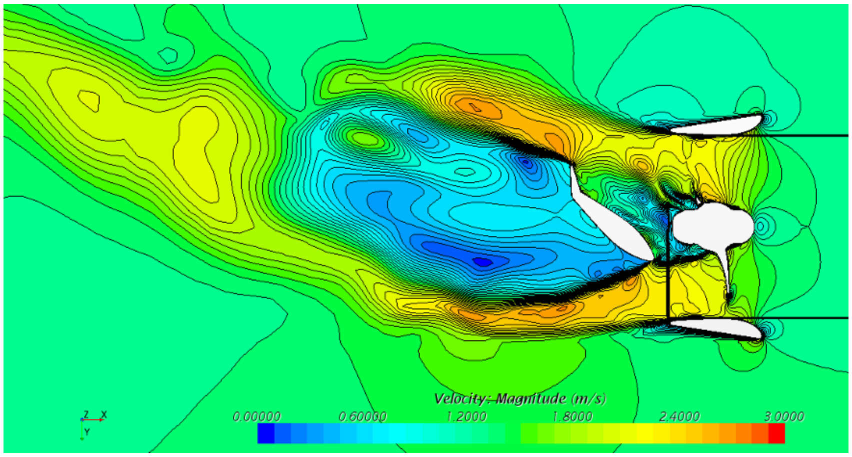


Figure 6. Velocity distribution of propeller-nozzle-rudder interaction

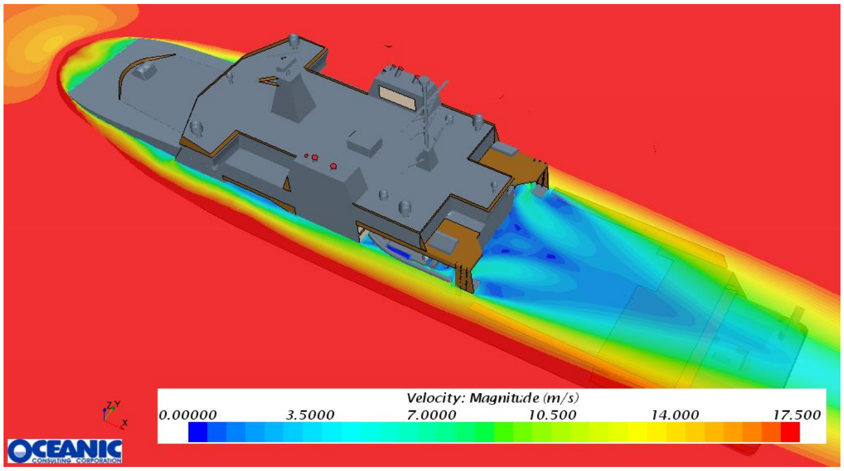


Figure 7. Velocity distribution of airflow past a vessel

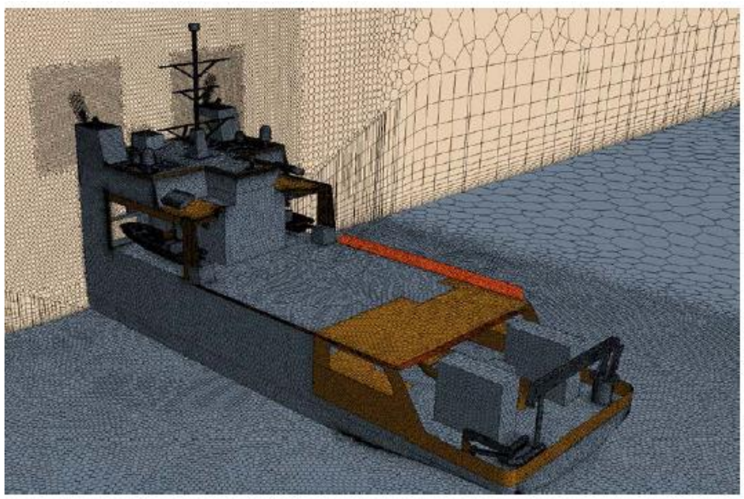


Figure 8. Meshes of airflow pass a vessel

2) Sloshing Prediction Inside a U-tube Tank

Natural gas storage and transportation on the ocean is another topic in a possible risk assessment. A sloshing motion in the tank will happen because of the vessel's vibration in the waves. The sloshing phenomenon is defined as high levels of liquid motion inside a container. The motion of the fluid in partially filled tanks may cause large structural loads if the period of ship motion is close to the natural period of the fluid inside the tank. On the other hand, the fluid motion in a tank may be used to damp vessel response in roll. That is the principle of the anti-roll tank. The approach to study these two sloshing flows is the same. Here we provide an example of sloshing prediction in a U-tube tank (Thanyamanta et al., 2013).

The U-tube tank geometry is shown in Figure 9. The U-tube tank partially filled with liquid was installed in a vessel at approximately amidships to reduce the vessel's roll motion (see Figure 10). A transfer pipe was added in order to match a designed property of the tank. In order to simulate liquid motion while the vessel was also moving in waves, an in-house code MOTSIM and a commercial software Flow-3D were fully coupled, where the vessel motion in waves was calculated by MOTSIM code, and then fluid motion with free surface in the U-tube tanks was simulated by Flow-3D which was in turn used to provide additional forces in the ship motion calculation in the next time step. Figure 12 shows an example of velocity magnitude contour obtained from Flow-3D. The figure shows velocity and free surface of the fluid in the tank as it flowed between the two side reservoirs. The locations where large dynamic pressure may be experienced are shown. It is useful information for designer of U-tube tanks and the same principles can be used to improve the safety of natural gas transportation. The nonlinearity in the free surface of the fluid in the tank induced by the translational and rotational motions of the ship can also be captured accurately by the code.

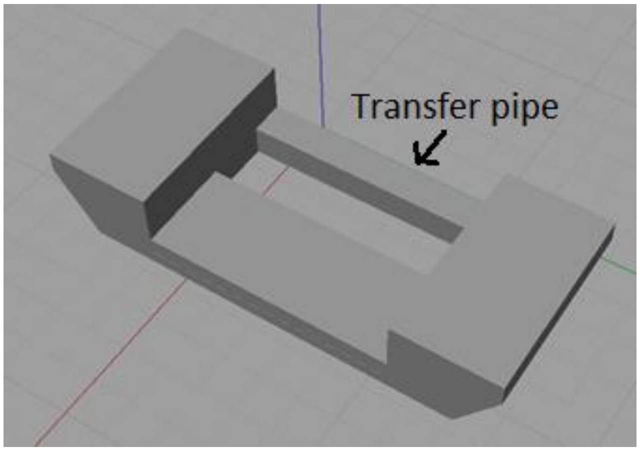


Figure 9. Geometry of a U-tube tank

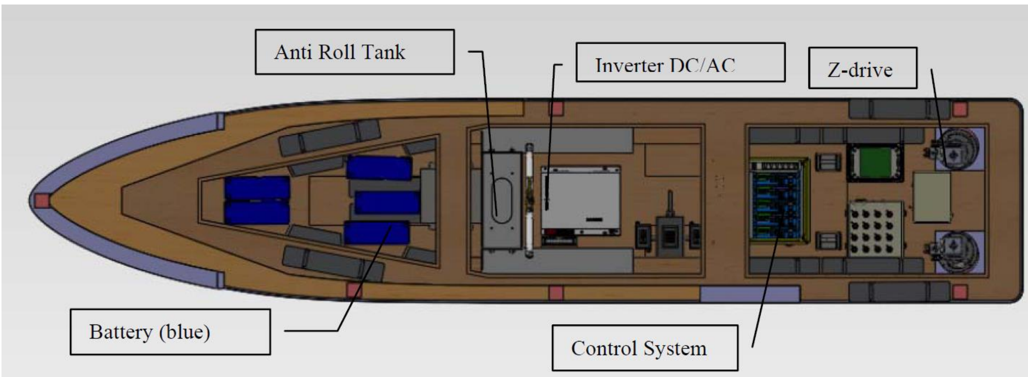


Figure 10. U-tube tank position in a vessel

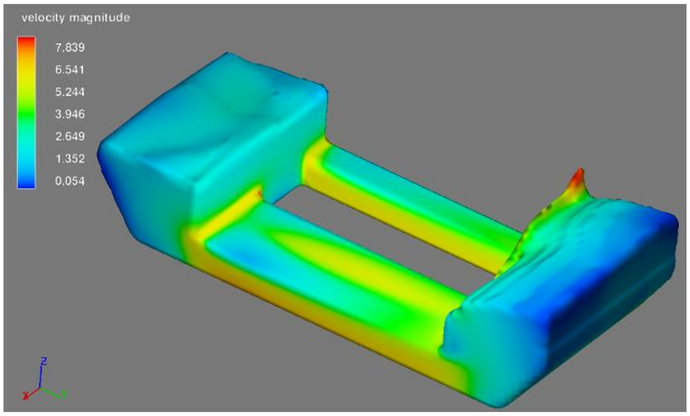


Figure 11. Velocity magnitude contour from U-tube tank simulation

3) Motion of Moored Structure in Waves and Visualization

The motion of ship or other large floating structures in waves (as shown in Figure 12) is also an important area in the offshore industry. Normally the theory of predicting ship motion is based on potential theory where the viscosity of fluid is ignored. In Oceanic an in-house code, MOTSIM, has been used to simulate seakeeping for many years.

MOTSIM is a non-linear time-domain seakeeping simulation program that can predict vessel motions in six degrees of freedom in almost any wave conditions. The seakeeping simulation produces a time history of the motion at the vessel's center of gravity as shown in Figure 13, and it also can provide wave force acting on the vessel.

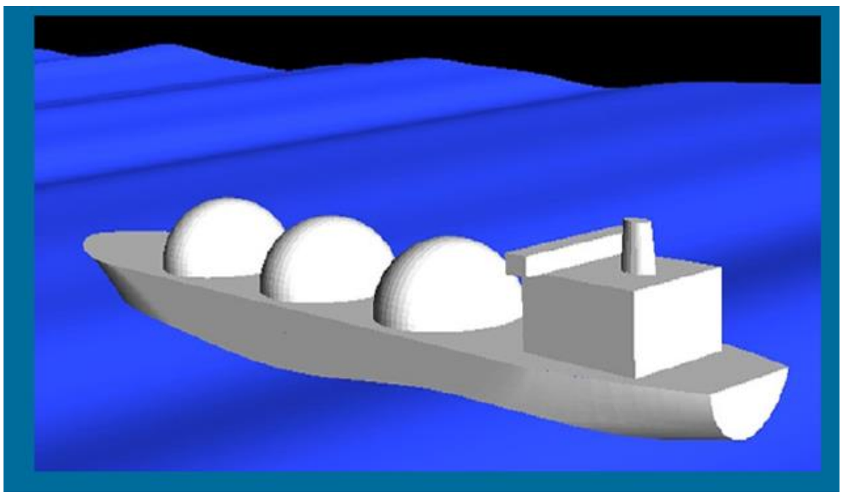


Figure 12. A ship motion in waves

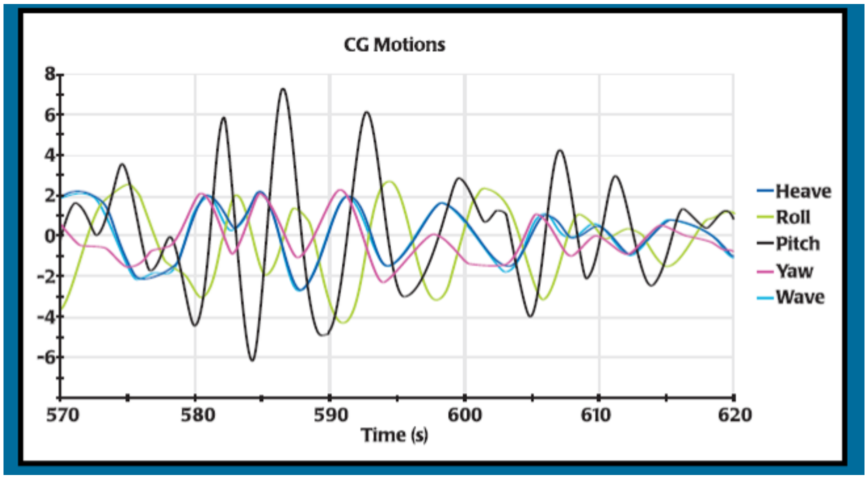


Figure 13. Time history of ship CG motion from MOTSIM simulation

A project to simulate the motion and resulting mooring loads of a moored FPSO in waves was recently conducted in Oceanic using MOTSIM code (see Figure 14). It not only simulates the FPSO movement in a range of waves, wind, and current, but also provides the mooring line tensions in the same time. The vessel's motion, wave pattern and mooring line tensions etc. were visualized and analyzed, which is extremely useful for a better understanding of the physical conditions. Figure 15 and Figure 16 show the interface of the visualizer and analyzer, respectively. The visualizer, developed in-house from open source software, including Ogre, Hydrax and Skyx, can playback a 3-dimensional view of the simulation (above and below water). The user can view the results in a movie-like format, and change viewing position during the play-back. The analyzer, for processing the time dependent responses generated by MOTSIM was developed using MATLAB, as a stand-alone analysis program. These methods of visualization and analysis can be applied to oil spill forecast investigation.

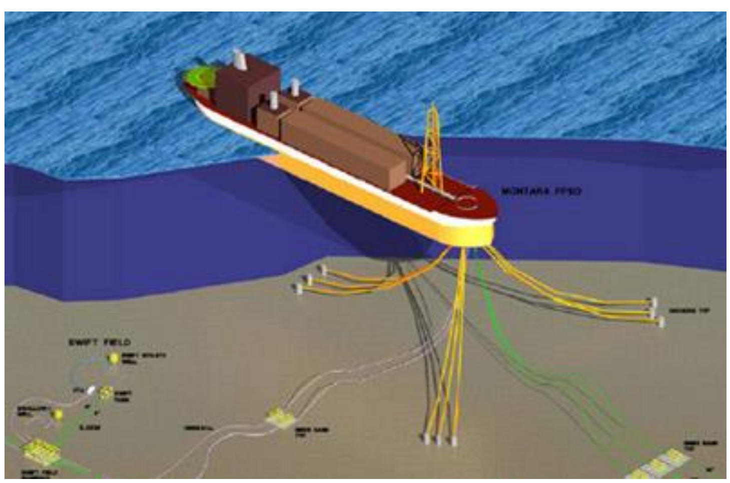


Figure 14. A turret moored FPSO

Figure 15. Screen shot of FPSO motion visualization

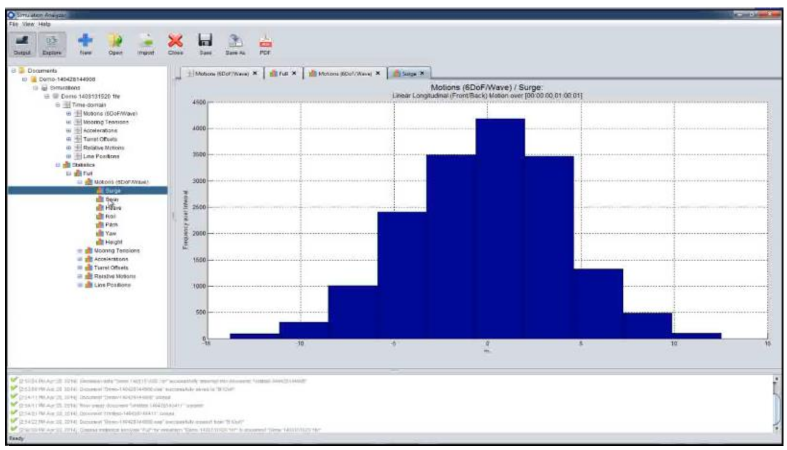


Figure 16. Interface of FPSO motion analyzer

Conclusion

This paper presents use of Computational Fluid Dynamics (CFD) at Oceanic Consulting Corporation (OCC) in recent years under support from the Atlantic Innovation Fund. This study includes numerical simulations of airflow or water flow over vessel or offshore structure, ship motion in waves and its visualization, and also liquid sloshing in a moving vessel by using commercial software and in-house codes. With helps of these tools it is possible and great opportunity for Oceanic to expand these studies and methods to some related areas such as oil spill forecast and response, oil and natural gas storage and transportation.

The overall goal of the work described in this paper is to increase the ability of the offshore oil industry to function safely and efficiently in harsh environments through a better understanding of structural responses in harsh environments. Offshore platforms working on the Grand Banks are subjected to higher levels of environmental forces than other areas of the world, and as a result may experience more extreme conditions such as waves breaking on the deck, and higher motion excursions due to the large waves. Wherever possible, the results of the simulations are checked against data from other sources, such as model experiments or full scale measurements.

Acknowledgement

This work was carried out at Oceanic as part of a project which was supported by the Atlantic Innovation Fund, administered by the Atlantic Canada Opportunities Agency, with additional financial support from Husky Energy Canada. The support from these two organizations is gratefully acknowledged.

The author would also like to thank colleagues Mohammed (Shameem) Islam, James McRoberts and Randy Cheema for providing the detailed results of numerical simulation and visualization.

References

Gaillarde, G. (2010). Shipmo: Motions & Seakeeping of Ships. www.marin.nl/web/file?uuid=28a7d199-70e9-4768-b4e4...88fc.

Gong, I.Y. (1993a). “Development of maneuvering simulation program: Part I – mathematical models for maneuvering motions in deep/shallow water with wind/current effects”, Institute for Marine Dynamics, Report No. IR-1993-09, 67 pages.

Gong, I.Y. (1993b). “Development of maneuvering simulation program: Part II – mathematical models for low speed maneuvering motions in restricted water area”, Institute for Marine Dynamics, Report No. IR-1993-11, 59 pages.

HydroStar for Experts User Manual, Research Department BUREAU VERITAS, March 2011.

INTERA, (1986). “DECICE theoretical manual”, Denver, Colorado, INTERA Information Technologies Environmental Division.

Pawlowski, J. and Bass, D. (1991). "A theoretical and numerical model of ship motion in heavy seas", SNAME Transaction, Vol.9, pp. 319-352.

Thanyamanta, W., Bass, D. and Molyneux, D. (2013). Prediction of sloshing effects using a coupled non-linear seakeeping and CFD code, OMAE2013, June 9-14, Nantes, France.