Journal of Multidisciplinary Engineering Science and Technology (JMEST)
ISSN: 3159-0040
Vol. 2 Issue 10, October - 2015

A First Approach On Positioning DIFIS System For Containing Oil And Oil-Gas Leaks

Dimitrios-Periklis A. Giannoulis
Mechanical Engineering and Aeronautics Dept.
University of Patras
Patras, Greece
giannoulis@mech.upatras.gr

Dionissios P. Margaris
Mechanical Engineering and Aeronautics Dept.
University of Patras
Patras, Greece
margaris@mech.upatras.gr

***Abstract*—Double Inverted Funnel for Intervention on Ship Wrecks (DIFIS) is a system designed for containing leaking oil from ship wrecks. The scope is to study its applicability as an offshore blowout intervention system where the conditions are completely different. A first step in this approach is to study the behavior of leaking oil and gas under seawater current conditions in order to re-examine system's positioning. Parametric numerical simulations of leaking oil and oil-methane mixtures are carried out for different seawater velocities before and after the positioning of DIFIS system above the leaking point and Dome's positioning and dimensions are proposed.**

Keywords—oil-well blowout; DIFIS; sea-water current; dome; oil-gas leak;

I. INTRODUCTION

Oil spills at sea due to offshore oil-well blowouts can lead to severe damage to the environment and the marine wildlife for generations. More often, oil spill cleanup efforts are insufficient to eliminate the pollution threat. As part of the effort to examine the applicability of DIFIS system to eliminate pollution threat from oil well blowouts, the positioning of the dome and its dimensions are examined, taking into account the rapid deployment needed, the physics of the plume formed and the possible sea-water currents at the region that can drift away the leaking mixture.

The DIFIS system, Fig. 1, has been initially designed, as mentioned earlier, for intervention on ship wrecks and has been studied on its hydrodynamics and structural properties [1-2] as well as from the internal point of view, by modelling the two phase oil-sea water buoyancy driven flow formed for various occasions [3-9].

During the recent years, a number of studies dealt with oil spill prevention and clean-up methods have been carried out. Abes [10] presented a safety and loss management system template that may be used for the full life cycle of a pipeline, from concept to abandonment.

Feng et al. [11] introduced the basic principles, process and methods of Quantitative Risk Assessment technology used to determine the failure scenarios of the facilities, estimate the possibilities of leakage failures, and calculate the consequences of failures and damages. Lee et al. [12] investigated experimentally and numerically the effectiveness of two oil fences deployed in tandem to maximize the containment of oil while Lee and Kang [13-14] studied the degradation of the effectiveness as an oil fence undergoes wave-excited motion and deformations due to sea currents.

The global demand for fossil fuels increases rapidly as indicated by Tanning [15] indicating the increased possibility of an accident which must not lead for another time to severe damage for the environment and the marine wildlife.

Li et al. [16] modeled submarine oil spills to forecast oil's trajectory under the effect of currents and waves.

In the present study, various cases are examined through 2D numerical simulations including leaking oil and oil-methane mixture for current velocities of 0.26 m/s and 1 m/s.

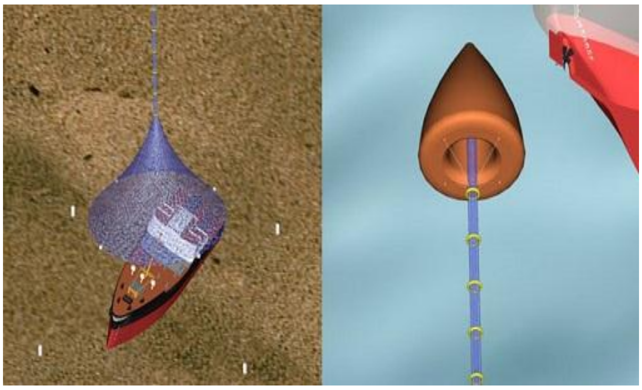

Fig. 1. DIFIS system

II. COMPUTATIONAL DETAILS

Computational fluid dynamics analysis is carried out with the commercial software package Fluent [17], which has been widely used in the field of fluid mechanics and the pre-processor Gambit 2.2.30 [18] is used for the grid formation. Cells of 4 cm width are used resulting in a total of 250000 quadrangles for the seabed, and 177400 quadrangles for seabed with DIFIS system, Fig. 2 and 3.

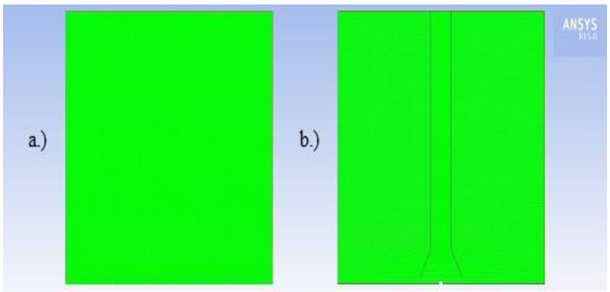

Fig. 2. a.) *Computational domain for seabed*
b.) *Computational domain with DIFIS system*

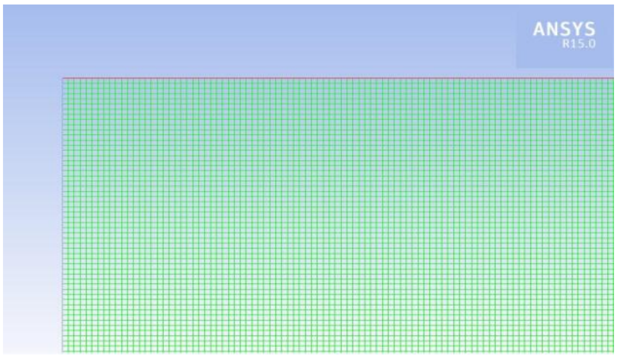

Fig. 3. *Closer view of the computational domain*

A. Turbulence Model

The turbulence kinetic energy, k, and its dissipation rate, ε, are obtained from the following transport equations:

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}\left(a_k \mu_{eff}\frac{\partial k}{\partial x_j}\right)+G_k+G_b-\rho\varepsilon-Y_M \tag{1}$$

$$\frac{\partial}{\partial t}(\rho \varepsilon)+\frac{\partial}{\partial x_i}(\rho \varepsilon u_i)=\frac{\partial}{\partial x_j}\left(a_\varepsilon \mu_{eff}\frac{\partial \varepsilon}{\partial x_j}\right)+C_{1\varepsilon}\frac{\varepsilon}{k}(G_k+C_{3\varepsilon}G_b)-C^*_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \tag{2}$$

where α_k and α_ε are the inverse effective Prandtl numbers for k and ε , G_k is the generation of turbulence kinetic energy due to the mean velocity gradients, G_b represents the generation of turbulence kinetic energy due to buoyancy, Y_M is the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate and $C^*_{2\varepsilon}$ is given by:

$$C^*_{2\varepsilon}\equiv C_{2\varepsilon}+\frac{C_\mu \eta^3(1-\eta/\eta_0)}{1+\beta\eta^3} \tag{3}$$

where $\eta_0=4.38$, $\beta=0.012$ and $S\equiv 2\sqrt{S_{ij}S_{ij}}$ with mean strain rate $S_{ij}=\frac{1}{2}\left(\frac{\partial u_j}{\partial x_i}+\frac{\partial u_i}{\partial x_j}\right)$.

Turbulence viscosity for low-Reynolds numbers is given by the following differential equation:

$$d\left(\frac{\rho^2 k}{\sqrt{\varepsilon\mu}}\right)=1.72\frac{\hat{v}}{\sqrt{\hat{v}^3-1+C_v}}d\hat{v} \tag{4}$$

where

$$\hat{v}=\mu_{eff}/\mu \tag{5}$$

which for high-Reynolds numbers gives:

$$\mu_t=\rho C_\mu\frac{k^2}{\varepsilon} \tag{6}$$

Model constants $C_{1\varepsilon}$, $C_{2\varepsilon}$ and C_v are:

$C_{1\varepsilon}=1.42$, $C_v\approx 100$ and $C_{2\varepsilon}=1.68$.

B. Multiphase Model

For the pth phase, the continuity equation for the volume fraction has the following form:

$$\frac{1}{\rho_p}\left[\frac{\partial}{\partial t}(\alpha_p\rho_p)+\nabla\cdot(\alpha_p\rho_p\vec{u_p})=S_{\alpha_p}+\sum_{p=1}^{n}(\dot{m}_{pq}-\dot{m}_{qp})\right] \tag{7}$$

where ρ_p is the density of the pth fluid. Also m_{pq} is the mass transfer from phase p to phase q and m_{pq} is the mass transfer from phase q to phase p. This volume fraction equation will be solved for the secondary phase. It will not be solved for the primary phase. The primary-phase volume fraction will be calculated based on the following constraint:

$$\sum_{p=1}^{n}\alpha_p=1 \tag{8}$$

The momentum equation is as follow:

$$\frac{\partial}{\partial t}(\rho\vec{u})+\nabla\cdot(\rho\vec{u}\vec{u})=-\nabla p+\nabla\cdot\left[\mu\left(\nabla\vec{u}+\nabla\vec{u}^T\right)\right]+\rho\vec{g} \tag{9}$$

C. Solution Methods and Controls

Geo-Reconstruct scheme is best suited for volume fraction, PISO algorithm is used for the Pressure-Velocity coupling and PRESTO interpolation scheme is used for pressure since gravity is the predominant force acting on the flow. Every other spatial discretization scheme is second order for precision issues while for the accuracy of the solutions, a value of 10^{-4} is used for all residual terms.

Leaking point depth is 1500 m, pressure is 150 bars and temperature 5^{o}C. The working fluid is sea-water with density ρ_w=1030 kg.m^{-3} and viscosity μ_w=155E-05 kg.m^{-1}s^{-1}. Secondary phases are GOM crude oil, with density ρ_o=891 kg.m^{-3} and viscosity μ_o=0.073 kg.m^{-1}s^{-1} and methane with density ρ_a = 133.4 kg.m^{-3} and viscosity μ_a = 1.736E-05 kg.m^{-1}.s^{-1}. Surface tension between oil and sea-water is $\sigma_{o/w}$=0.025 N/m, between methane and sea-water is $\sigma_{m/w}$=0.074 N/m and between oil and methane $\sigma_{o/m}$=0.031 N/m.

As the dome and riser materials are undefined, a no slip boundary condition is the most appropriate. In addition, other boundary conditions are 'wall', 'pressure outlet' and 'velocity inlet'.

III. RESULTS AND DISCUSSION

Cases examined through 2D numerical simulations include leaking oil and oil-methane mixture at flow rates of 30000 bpd (barrels per day) and 60000 bpd for constant sea-water current velocities of 0.26 m/s and 1 m/s.

Leaking oil seems to rise up towards the surface as a continuous column. It is generally known that leaking mixture breaks up into droplets and bubbles a few meters above the discharge point but RANS turbulence models cannot capture weak flow instabilities which are responsible for these formations. However, in this study, RNG k-ε model can cover with reasonable accuracy the examined behavior of the flow under sea-water currents in order to make an estimate about the dome positioning.

Oil released in the seabed is driven into the water column as a jet due to the momentum of the discharge. While buoyancy is dominant for relatively low current velocity, with increasing velocity it is obvious that the influence of the current becomes important.

With sea-water current velocity at 0.26 m/s and leaking flow rate of 30000 bpd, Fig. 4, as the dynamic character of the jet gradually decreases, leaking oil is dispersed by coming water current forming small oil bubbles and particles which are drifted for about 10m, 4m above the leaking point. In the extreme case of 1m/s sea current, Fig. 5, oil column attaches to the sea floor before breaking into bubbles which could continue rising, increasing the difficulties for the recovery and clean up efforts.

For leaking flow rate of 60000 bpd the situation is similar for the case of 1m/s sea-water current velocity, Fig. 6. However, for lower current velocity, 0.26 m/s, oil remains and rises as a jet column that is drifted for 5m due to the increased momentum of the discharge, Fig. 7.

Situation is not very different for the leaking mixture of oil and gas. At current velocity of 1 m/s, oil and gas particles formed are drifted away immediately above the leaking point, Fig. 8, 9, 10 and 11. The first major difference to the previous examined scenario is that the form and the drift of the leaking mixture are not seriously affected by the leaking flow rate while the second difference is that it is not observed the previously referred case of the leaking jet attached to the sea floor.

Oil and gas bubbles and particles formed above the leaking point influence the current increasing its velocity near the leaking point in all cases and giving a rise in the mixture velocity which also increases due to buoyancy as well.

From the above results for the behaviour of leaking oil and oil-gas mixture under different sea-water current velocities, it is obvious that the initial configuration of the DIFIS system and specifically of the dome positioning, is not appropriate. The possibility that leaking mixture escapes and cannot be captured by the dome is increased as it is positioned about 20 m above the accident point. In addition, its size is appropriate for containing the leaks from a ship wreck but is not necessary in the case of an oil well blowout because there is only one leaking point. As a result, it is proposed that its diameter and height could be reduced to 4 m and 2 m respectively. Taking these dimensions into account, it is examined the flow behaviour after positioning the hole system just 0.5 m above the leaking point. The cases examined include 1 m/s sea-water current velocity and 30000 bpd leaking flow rate for oil, Fig. 12 and oil-gas mixture, Fig. 13 and 14, in order to have relatively low discharge momentum and eliminate the buoyancy dominance region.

It is observed that, even if there are some small quantities that may escape, the flow stays almost vertical and seems unaffected by sea-water currents.

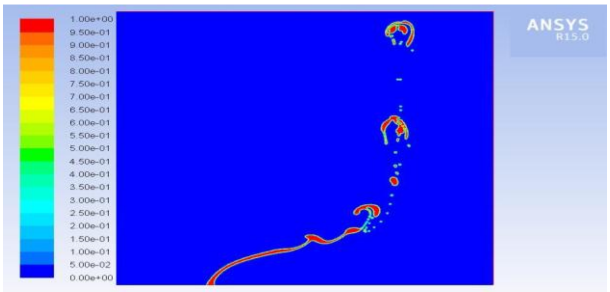

Fig. 4. *Oil leak, 30000 bpd, (0.26 m/s current velocity)*

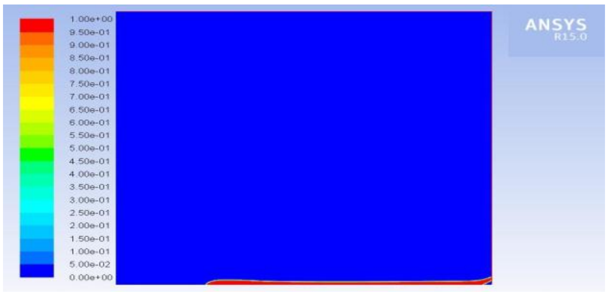

Fig. 5. *Oil leak (30000 bpd, 1 m/s current velocity)*

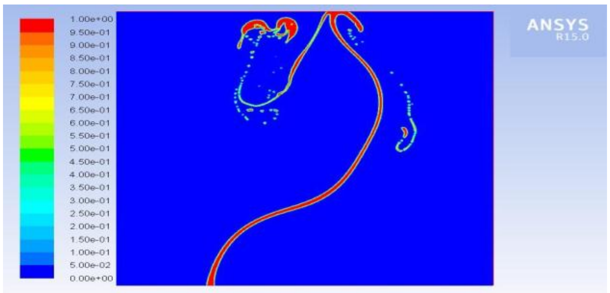

Fig. 6. *Oil leak (60000 bpd, 0.26 m/s current velocity)*

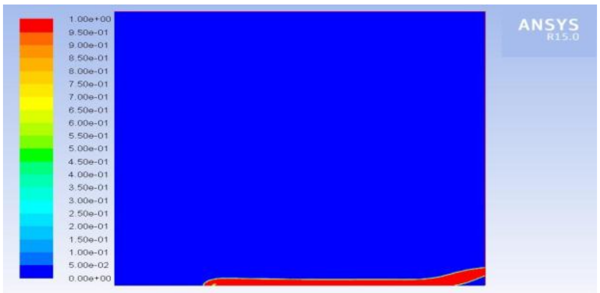

Fig. 7. *Oil leak (60000 bpd, 1 m/s current velocity)*

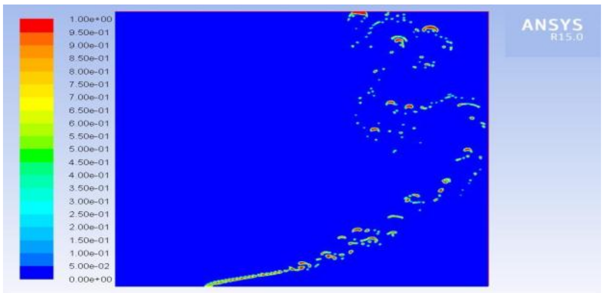

Fig. 8. *Oil-methane leak, methane (30000 bpd, 1 m/s current velocity)*

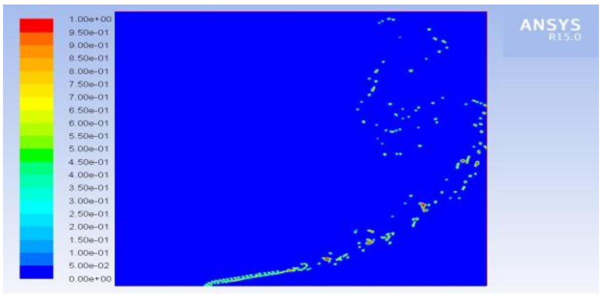

Fig. 9. *Oil-methane leak, oil (30000 bpd, 1 m/s current velocity)*

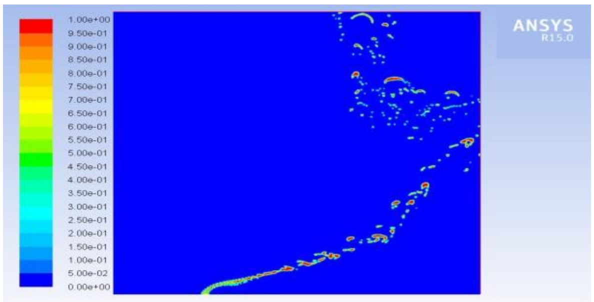

Fig. 10. *Oil-methane leak, methane (60000 bpd, 1 m/s current velocity)*

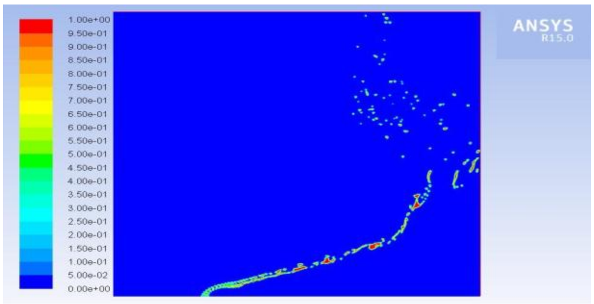

Fig. 11. *Oil-methane leak, oil (60000 bpd, 1 m/s current velocity)*

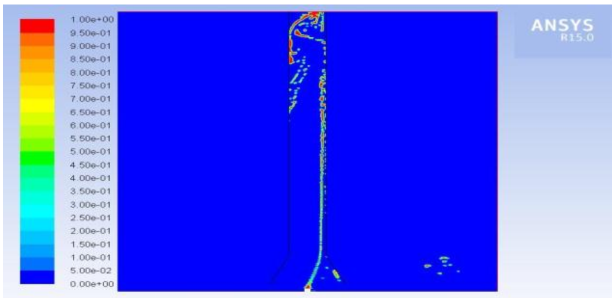

Fig. 12. *Oil leak with DIFIS system (30000 bpd, 1 m/s current velocity)*

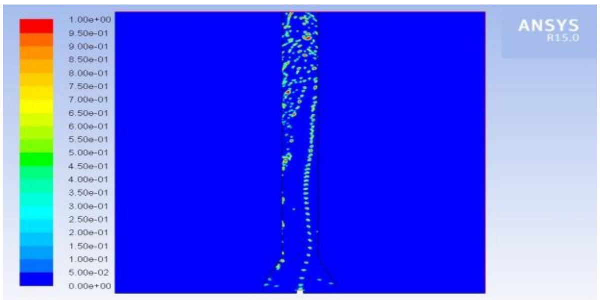

Fig. 13. *Oil-methane leak with DIFIS system, methane (30000 bpd,1 m/s current velocity)*

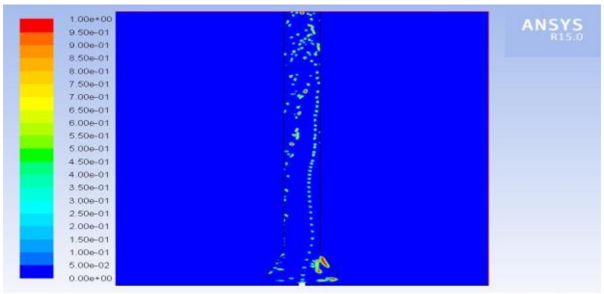

Fig. 14. *Oil-methane leak with DIFIS system, oil (30000 bpd, 1 m/s current velocity)*

IV. CONCLUSION

Maritime accidents leading to major environmental pollution due to releases of crude oil from offshore platforms and drilling rigs occur regularly. This study aims at the simulation of the flow of oil and oil-methane mixture under seawater conditions in order to examine the applicability of DIFIS system in subsurface oil-well blowouts. It is concluded that after positioning the system at lower height above the leaking point, the flow is almost unaffected by seawater currents and Dome's diameter could be reduced as there is one leaking point.

REFERENCES

[1] F. Andritsos, P. A. Konstantinopoulos, K. J. Charatsis, "Recuperation of oil trapped in ship wrecks: the DIFIS concept", Proc. SSE07, vol. Track A, pp. 1-9, 2007.

[2] F. Andritsos, H. Cozijn, "An innovative method for containing offshore oil well blow-outs", Proc. ASME 30th International Conference on Ocean, Offshore and Arctic Engineering, OMAE 2011-49110, pp. 1-10, 2011.

[3] D. P. A. Giannoulis, D. P. Margaris, "Oil leakages caused by maritime accidents: computational study of a recuperating method based on buoyancy driven flows", Proc. 5th International Conference from Scientific Computing to Computational Engineering, 2012,Vol. II, pp. 368-375.

[4] D. P. A. Giannoulis, D. P. Margaris, "Computational study of the two-phase flow formed within the vertical pipe of a system designed to remove leaking oil from maritime accidents", International Review of Mechanical Engineering, Vol. 6 N. 7, pp. 1694-1699, 2012.

[5] D. P. A. Giannoulis, D. P. Margaris, "Numerical simulation of leaking oil collection from a ship wreck", Proc. 5th International Conference on Experiments/Process/System Modeling/Simulation/Optimization, 2013, Vol. II, pp. 401-409.

[6] D. P. A. Giannoulis, D. P. Margaris, "Numerical simulation of the three-phase flow formed within the riser tube of a system designed to remove

leaking oil from maritime accidents”, International Review of Mechanical Engineering, Vol. 8 N. 1, pp. 94-99, 2014.

[7] D. P. A. Giannoulis, D. P. Margaris, “3D CFD Simulation of an oil spill prevention system based on buoyancy driven flows”, Proc. 6th International Conference from Scientific Computing to Computational Engineering, 2014, Vol. I, pp. 1-12.

[8] D. P. A. Giannoulis, D. P. Margaris, “Assessment of the behavior, of oil trapped in the riser tube of an oil spill prevention system based on buoyancy driven flow”, International Review of Mechanical Engineering, Vol. 8 N. 6, pp. 1110-1115, 2014.

[9] D. P. A. Giannoulis, D. P. Margaris, “Numerical simulation of the flow within DIFIS system during the containing of an offshore oil well blowout”, Proc. 6th International Conference on Experiments/Process/System Modeling/Simulation/Optimization, 2015, Vol. II, pp. 288-298.

[10] J. Abes, “Safety and loss management for pipelines”, Journal of Pressure Vessel Technology, vol. 130, 2008.

[11] W. Feng, X. Xiang, G. Jia, L. Dai, Y. Gu, X. Xang, Q. Feng, L. Zhou, “Applying the quantitative risk assesment (QRA) to improve safety management of oil and gas pipeline station in China”, Proceedings ASME: Pipelining in Northern and Offshore Environments, Strain-Based Design; Risk and Reliability; Standards and Regulations, 24 September, Vol. 4, pp. 505-512, 2012.

[12] C.M. Lee, K.H. Kang, N.S. Cho, “Trapping of leaked oil with tandem oil fences with langrangian analysis of oil droplet motion”, Journal of Offshore Mechanics and Arctic Engineering, Vol. 120, pp. 50-55, 1998.

[13] C.M. Lee, K.H. Kang, “Development of optimum oil fences in currents and waves”, Annual Report of Advanced Fluids Engineering Research Center, Series AFR-94-FGH, pp. 7-41, 1995.

[14] C.M. Lee, K.H. Kang, “Draft loss of a flexible oil fence in currents due to deformation”, Annual Report of Advanced Fluids Engineering Research Center, Series AFR-95-FGH, pp. 7-36, 1995.

[15] L. Tanning, T. Tanning, “Analysis of the fossil fuels situation of new members of the European Union to 2014”, Journal of Multidisciplinary Engineering Science and Technology, vol. 2, Issue 1, pp. 345-355, January 2015.

[16] W. Li, Y. Pang, J. Lin, X. Liang, “Computational modeling of submarine oil spill with current and wave by FLUENT”, Research Journal of Applied Sciences, Engineering and Technology, Vol. 5, pp. 5077-5082, 2013.

[17] Fluent Inc., Fluent 15 Documentation-User’s Guide, 2013.

[18] CFD Pre-processor, Gambit 2.3 Documentation, 2004.