International Journal of Mechanical, Industrial and Aerospace Sciences
ISSN: 2517-9950
Vol:11, No:11, 2017

Numerical Simulation of the Flow Channel in the Curved Plane Oil Skimmer

Xing Feng, Yuanbin Li

***Abstract*—**Oil spills at sea can cause severe marine environmental damage, including bringing huge hazards to living resources and human beings. In situ burning or chemical dispersant methods can be used to handle the oil spills sometimes, but these approaches will bring secondary pollution and fail in some situations. Oil recovery techniques have also been developed to recover oil using oil skimmer equipment installed on ships, while the hydrodynamic process of the oil flowing through the oil skimmer is very complicated and important for evaluating the recovery efficiency. Based on this, a two-dimensional numerical simulation platform for simulating the hydrodynamic process of the oil flowing through the oil skimmer is established based on the Navier-Stokes equations for viscous, incompressible fluid. Finally, the influence of the design of the flow channel in the curved plane oil skimmer on the hydrodynamic process of the oil flowing through the oil skimmer is investigated based on the established simulation platform.

***Keywords*—**Curved plane oil skimmer, flow channel, CFD, VOF.

I. INTRODUCTION

In recent years, accidents such as the Gulf of Mexico oil spill and the Dalian “7.16” oil spill have the potential to cause significant damage to the marine ecological environment. Many technologies have been developed to clean-up oil spills [1]-[3], including in situ burning, using chemical dispersants to break-up the oil and to speed its natural biodegradation, and the use of mechanical devices. In situ burning or chemical dispersant methods can be used in many situations; however, these methods present many disadvantages, e.g. the in situ burning may fail in slightly rough seas and chemical dispersant method can be highly toxic. Mechanical devices such as oil containment booms have been employed in calm seas to confine spilled oil which can be subsequently recovered. The concept of oil-containment boom was studied frequently numerically based on the finite volume method (FVM) and finite element method (FEM) [4]-[6]. Oil skimmers are one of the most widely used equipment to recover the spilled oil in marine or inland waters. There are many types of oil skimmers, such as curved plan oil skimmer, weir skimmer, belt skimmer, vacuum suction skimmer and so on, most of which have the disadvantage of low recovery efficiency, being easily influenced by the ambient environment such as the wave, current, and wind. Many of them have been studied with an objective to achieve high recovery efficiency and ease of operation, such as elimination units for marine oil pollution (EUMOP) [7] and seaway independent oil skimming system (SOS) [8] for both calm water and rough seas. The curved plane oil skimmers are commonly used due to the capacity of anti-wind, wave and current. Fig. 1 shows the prototype and the schematic diagram of the curved plane oil skimmer system. The sealed chamber in the oil skimmer is fulfilled with water and has a discharge port both at the top side and back side which is connected to the pumps. At the front side, there is a curved plane flow channel connected with the sealed chamber which is consisted of the upper layer and lower layer curved plane and the chamber side walls. Note that the principle of the oil recovery for the curved plane oil skimmer is rather simple. Initially, the pump connected with the discharge port at the back side is turned on to pump the water out of the chamber. Due to the low pressure in the chamber, a mixture of water and oil enters into the chamber through the curved plane flow channel. Under the diversion of the curved plane, the oil slick will be disturbed to dispersed oil droplets at the end of the channel. After entering into the sealed chamber, the dispersed oil droplets will re-gather at the upper part of the chamber to form the oil slick due to the combined effects of the buoyancy, viscous resistance, and the water current. When the thickness of the re-gathered oil slick reaches a certain value, the pump connected with the top discharge port will be turned on to deliver the oil slick.

Fig. 1 The prototype of the curved plane oil skimmer system

Despite their being widely used, the curved plane oil skimmers are a long way from being fully efficient and feature multiple failures. And the hydrodynamic aspects of the recovery process are very complicated since it involves multiphase and multiscale moving interfaces, including oil, water and moving interface of oil slick, oil droplets of different scales. In order to optimize the geometry of curved plane oil skimmer for improving the recovery efficiency and capacity of the curved plane oil skimmers to protect the ocean environment after oil spills, majority of studies are carried out including

Xing Feng is with the Dalian Maritime University, Dalian, DL 116026 China (corresponding author, phone: +8618940926190; fax: +8641184729338; e-mail: 530546168@ 163.com).

Yuanbin Li is with the Dalian Maritime University, Dalian, DL 116026 China (e-mail530546168@qq.com).

physical experimental and numerical approaches. As reported in the previous research, the curved plane flow channel played an important role in the hydrodynamic aspects of the recovery process, and the upper layer (arc BC) affected the distribution of the oil slick in the sealed chamber and recovery efficiency hugely. However, there is insufficient information about the influence of the flow channel pattern on separation of the oil and water in the curved plane oil skimmer. As pointed out by the previous research, there are three kinds of the flow channel according to the shape of the upper layer (arc BC) as shown in Fig. 2 (a) tapered curved plane flow channel, (b) parallel curved plane flow channel, (c) gradually expanding curved plane flow channel), and the gradually expanding curved plane flow channel has the best recovery efficiency. Here, only the gradually expanding curved plane flow channel will be considered. Firstly, the 2-D numerical simulation platform for the simulation of the oil recovery process in the curved plane oil skimmer will be established. The recovery process is simplified into a two-phase flow problem involving oil and water. The FLUENT 13.0 is adopted for the simulations. The volume of fluid method (VOF) method is employed to capture the moving surfaces between the fluid phases. Studies were also extended to investigate the influence of the geometry of the gradually expanding curved plane flow channel on the velocity and oil slick distribution in the sealed chamber.

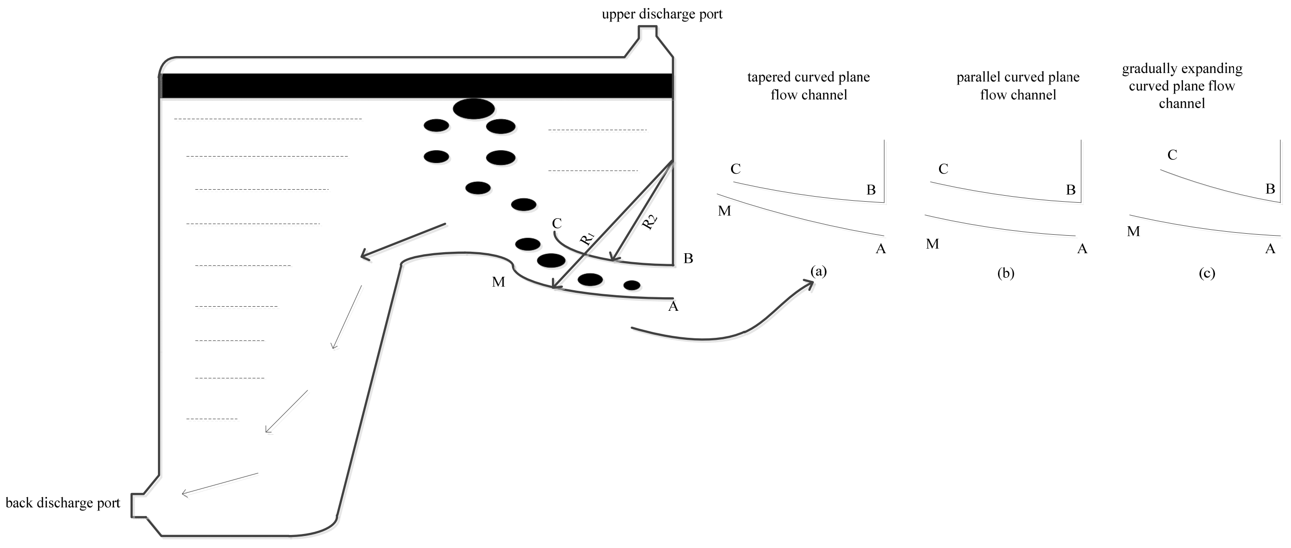


Fig. 2 Schematic diagram of the curved plane oil skimmer

II. NUMERICAL EXPERIMENTAL SET-UP

A. Governing Equations

The incompressible fluid motion (oil and water) can be described by the mass conservation equation, momentum conservation equations and RANS equations [9]-[11].

$$\frac{\partial (u_i)}{\partial x_i} = 0. \tag{1}$$

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(\rho u_i u_j)}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right) - \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left(-\rho\overline{u_i' u_j'}\right) + \rho g_i. \tag{2}$$

in which t is time and ρ is the density, x_i denotes the Cartesian coordinates, u_i is an ensemble mean velocity component, p is the fluid pressure, μ is the dynamic viscosity, and g is the gravitational acceleration. $-\rho\overline{u_i' u_j'}$ is the Reynolds stress term. The solution about the Reynolds stress term can be found in the Manual Guide of FLUENT in detail [12].

The VOF method is used to track the oil-water surface. The main idea of VOF method is to define a function α to represent the fractional volume of water fluid:

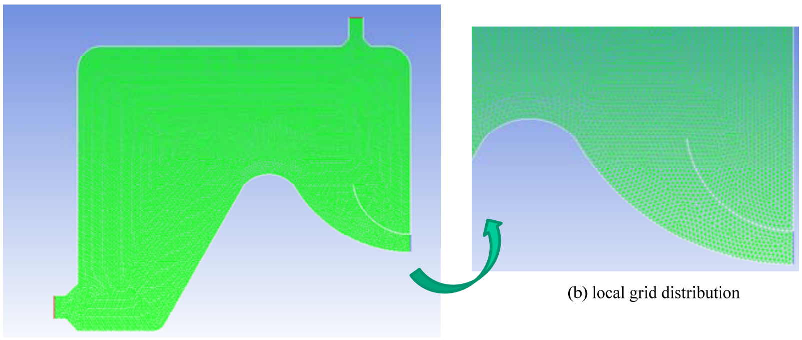
$$\frac{\partial \alpha}{\partial t} + \frac{\partial (u_i \alpha)}{\partial x_i} = 0. \tag{3}$$

$\alpha = 1$ indicates that the cell is full of water, while $\alpha = 0$ corresponds to a cell fully occupied by oil. Cells with value of $0 < \alpha < 1$ contain an oil-water free surface.

B. Mesh Generation

The commercial software FLUENT 13.0 has been used for the simulations. The computational zones were discretised by the structured grids using GAMBIT. It is well known that the accuracy of CFD results and the calculation time strongly depend on the type of the mesh and number of cells used. So, calculations have been done several times to find optimum mesh number. 20,000 meshes are suitable for the geometry used here to get correct results as shows in Fig. 3. In order to analyze the influence of the geometry of the gradually expanding curved plane flow channel on the velocity and oil slick distribution, the radius of curvature of the upper layer (arc

BC) are chosen to be $R_2 = 394mm$, $R_2 = 294mm$ and $R_2 = 244mm$. The mesh information can be found in Table I.


(b) local grid distribution

(a) computational domain grid distribution

Fig. 3 Computational domain and local grid distribution

TABLE I
MESH INFORMATION FOR DIFFERENT GEOMETRICAL MODEL

Case Number	R_2 (mm)	Minimum mesh volume(m^3)	Maximum mesh volume(m^3)	Cells	Nodes
1	394	2.229869×10^{-05}	7.312881×10^{-05}	28035	14341
2	294	2.442687×10^{-05}	7.353020×10^{-05}	28003	14320
3	244	2.161736×10^{-05}	7.840026×10^{-05}	28017	14324

C. *Numerical Solution Methods*

The CFD code, FLUENT 13.0, which has been proven to be viable for solving compressible and incompressible flows based on the two dimensional or three-dimensional N-S equations or RANS equations, is chosen as the base solver. In the following, only the numerical methods used here are briefly described. For turbulent flows, the FLUENT solver supports various turbulence models. When deciding the exact turbulence model, we find that the standard k-e model converged faster and gives better results than the other turbulence models. Therefore, the standard k-e model using a linear pressure-strain model is used in conjunction with the standard wall function method for a smooth wall for all the present simulations. With the above models, the FLUENT solver employing the FVM for the discretion of the governing equations on the staggered grid generally solves the RANS. The VOF employing a geometric reconstruction scheme is used to track free surface movement. Pressure, turbulent kinetic energy, turbulent dissipation rate and water volume fraction function are arranged at the center point of grid, fluid velocity components are arranged at the center point of corresponding grid boundary. The body force-weighted scheme was chosen for pressure interpolation and the second order upwind scheme for discretization of the momentum equation. In order to calculate convection and diffusion fluxes through the control volume faces, PISO algorithm is adopted for pressure-velocity coupling. The convergence criterion is set to 0.00001 for all simulations.

D. *Boundary Conditions and Initial Conditions*

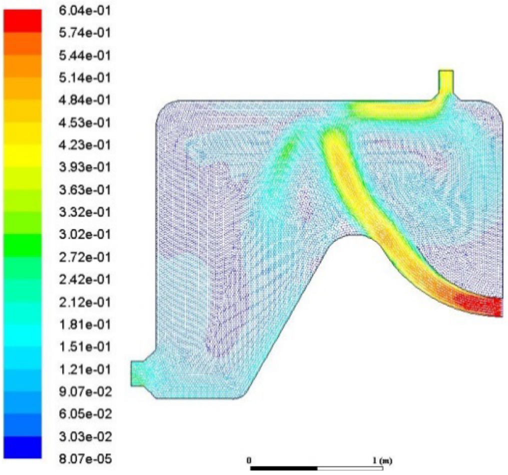
The no-slip boundary with the smooth wall function is set for all the inner walls of the sealed chamber and the curved plane flow channel. The velocity-inlet boundary is set for inlet-AB and the outflow boundary is set for the upper discharge port and the back discharge port. For the velocity-inlet boundary, it is reasonable to assume that the oil skimmer does not move and the mixture of water and oil flows into the oil skimmer, and the velocity is chosen to be 0.6 m/s. The fluid properties of the mixture are given in Table II. The volume fraction of oil is chosen to be 0.2.

TABLE II
FLUID PROPERTIES

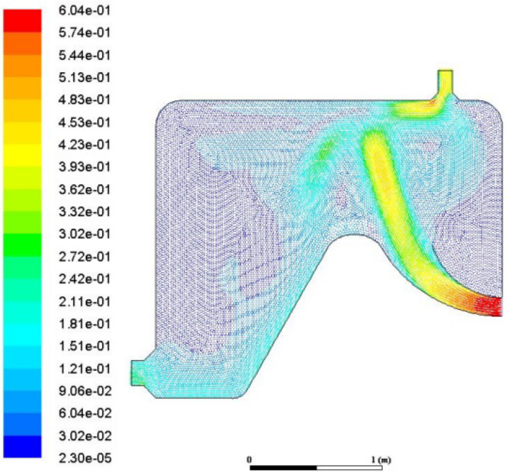
	Density(kg/m^3)	Viscosity(kg/m-s)
Oil	900	0.09
Water	1030	0.001003

III. NUMERICAL RESULTS AND DISCUSSION

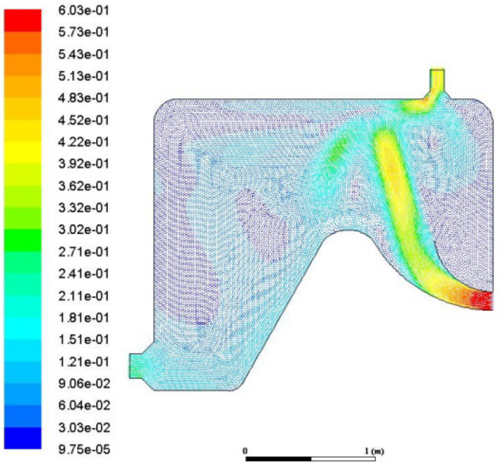
The snapshots of velocity patterns in the oil skimmer with different curved plane flow channel at computational time $t = 120s$ are displayed in Fig. 4. As shown in Fig. 4, the curved plane flow channel has the diversion effect on the mixture of the oil and water entering the inlet port AB. After peeling off the curved plane flow channel, the mixture will have certain vertical velocity component, and this vertical velocity component varies with the shape of the upper layer (arc BC) of the curved plane flow channel which is determined by the radius of curvature of the arc BC(R_2). The smaller the R_2, the larger the vertical velocity component, which indicates better oil-water separation due to the better floating up of the oil droplets.



(a) $R_2 = 394mm$



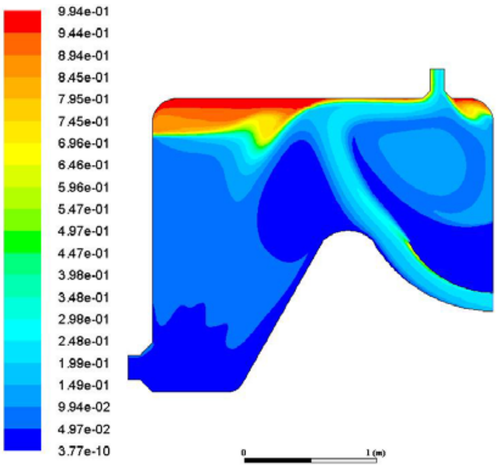
(b) $R_2 = 294mm$



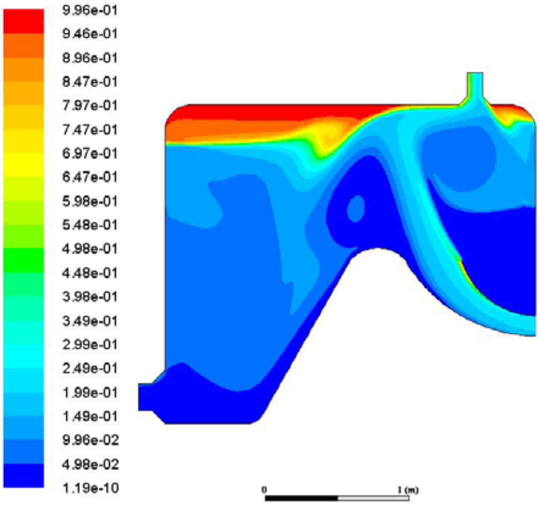
(c) $R_2 = 244mm$

Fig. 4 The snapshots of velocity patterns in the oil skimmer with different curved plane flow channel

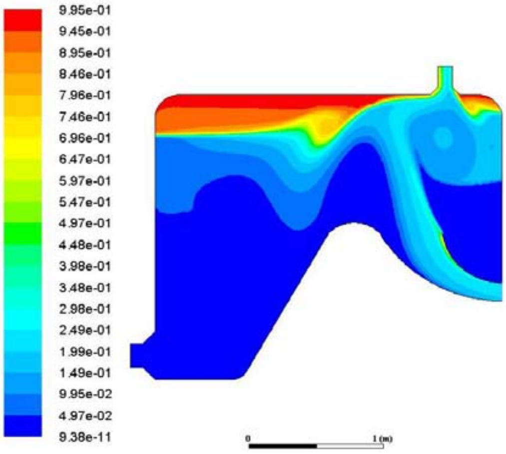
The snapshots of oil phase distribution in the oil skimmer with different curved plane flow channel at computational time $t = 120s$ are displayed in Fig. 5. As shown in Fig. 5, the oil-water will be separated in the sealed chamber after peeling off the curved plane flow channel. For $R_2 = 394mm$ as shown in Fig. 5 (a), the peeling off direction of the mixture bias towards the back side of the discharge port and certain amount of oil droplet will be discharged towards the back side of the discharge port. For $R_2 = 294mm$ as shown in Fig. 5 (b), the thickness of the floating oil slick re-gathered in the top of the sealed chamber is larger than that in the oil skimmer with $R_2 = 394mm$ although certain amount of oil droplet is moving towards the back side discharge port. For $R_2 = 244mm$ as shown in Fig. 5 (c), the thickness of the floating oil slick re-gathered in the top of the sealed chamber is largest and no oil droplets will be discharged towards the back side discharge port, which indicates the best recovery efficiency.



(a) $R_2 = 394mm$



(b) $R_2 = 294mm$



(c) $R_2 = 244mm$

Fig. 5 Oil phase volume fraction distribution contour map

IV. CONCLUSIONS

In this paper, the 2-D numerical simulation platform for the simulation of the flow channel in the curved plane oil skimmer is established based on the Navier-Stokes equations for viscous, incompressible fluid. Firstly, the Gambit software is chosen to discretise the computational model and to generate the structured grids. Then, the CFD code, FLUENT 13.0, is chosen as the base solver. The VOF model employing a geometric reconstruction scheme is used to track the free surface between the oil and water. The body force-weighted scheme is chosen for pressure interpolation and the second order upwind scheme for discretization of the momentum equation, PISO algorithm is adopted for pressure-velocity coupling. Finally, the effect of the shape of the upper layer of the curved plane flow channel on the recovery efficiency is analyzed. For certain ambient current velocity, the smaller the radius of curvature of the upper layer of the curved plane flow channel, the better the recovery efficiency. The quantitative analysis of the effect of the radius of curvature of the upper layer under different ambient current velocity will be carried out in the future studies.

ACKNOWLEDGMENT

This work was supported in part by the National Natural Science Foundation of China with Grant (Project nos.:51409032 and 51409031), the Fundamental Research Funds for the Central Universities (Project nos.: 3132017006, 3132016314) (China).

REFERENCES

[1] Abascal, A. J., Castanedo, S., Medina, R., and Liste, M., “Analysis of the Reliability of a Statistical Oil Spill Response Model,” Mar. Pollut. Bull., 60(11), pp. 2099–2110, 2010.

[2] Fraser, J. P., and Clark, L. M. C., “SOCK Skimmer-Performance and Field Tests,” J. Pet. Technol., 36(3), pp. 451–456, 1984.

[3] Leibovich, S., “Hydrodynamic Problems in Oil-Spill Control and Removal,” J. Pet. Technol., 29(3), pp. 311–324, 1977.

[4] FENG Xing, WU Wanqing, “Oil Containment by Floating Boom under Wave-current Coupling Conditions,” I&II. ICEEP2013, 2013.

[5] Azin Amini, Anton J. Schleiss, “Behavior of rigid and flexible oil barriers in the presence of waves,” Applied Ocean Research, vol.31, pp.186-196, 2009.

[6] G. Iglesias, A. Castro, J. A. Fraguela, “Artificial intelligence applied to floating boom behavior under waves and currents,” Ocean Engineering, vol.37, pp.1513-1521, 2010.

[7] Lemesle, P., Kakalis, N., Fritsch, D., and Turan, O., “Design of Monohull EUMOP Units to Clean Oil Spills,” Small Craft Conference, Bodrum, Turkey, pp. 331–366, 2006.

[8] Clauss, G., Abu-Amro, M., and Kosleck, S., “Numerical and Experimental Optimization of a Seaway Independent Oil Skimming System—SOS,” 16th International Offshore and Polar Engineering Conference, San Francisco, CA, pp. 495–502, 2006.

[9] D. Markus, F. Ferri, “Complementary numerical-experimental benchmarking for shape optimization and validation of structures subjected to wave and current forces,” Computers & Fluids ,118:, pp.69-88, 2015.

[10] Shaoyu Ni, Wei Qiu, “Hydrodynamic simulation and optimization of an oil skimmer,” Journal of Offshore Mechanics and Arctic Engineering, vol.137, pp.021301-1-021301-10, 2015.

[11] J. H. Ferziger, “Computational methods for fluid dynamics,” Springer, New York, 2002.

[12] ANSYS/FLUENT, Fluent user’s guide (Version 13.0). Canonsburg (WA): ANSYS Inc, 2012.

Xing Feng was born in 1985in Linchuan, Jiangxi Province, China. He received his Doctor degree in Dalian Maritime University. After his ten years studying in marine engineering, he worked as a teacher at DMU. Now, he is conducting research about CFD analysis of the floating structure in ocean waves and currents.