World Academy of Science, Engineering and Technology
International Journal of Mechanical and Mechatronics Engineering
Vol:9, No:5, 2015

Numerical Simulation of Free Surface Water Wave for the Flow around NACA 0012 Hydrofoil and Wigley Hull Using VOF Method

Saadia Adjali, Omar Imine, Mohammed Aounallah, Mustapha Belkadi

Abstract**—**Steady three-dimensional and two free surface waves generated by moving bodies are presented, the flow problem to be simulated is rich in complexity and poses many modeling challenges because of the existence of breaking waves around the ship hull, and because of the interaction of the two-phase flow with the turbulent boundary layer. The results of several simulations are reported. The first study was performed for NACA0012 of hydrofoil with different meshes, this section is analyzed at h/c= 1, 0345 for 2D. In the second simulation a mathematically defined Wigley hull form is used to investigate the application of a commercial CFD code in prediction of the total resistance and its components from tangential and normal forces on the hull wetted surface. The computed resistance and wave profiles are used to estimate the coefficient of the total resistance for Wigley hull advancing in calm water under steady conditions. The commercial CFD software FLUENT version 12 is used for the computations in the present study. The calculated grid is established using the code computer GAMBIT 2.3.26. The shear stress k-ωSST model is used for turbulence modeling and the volume of fluid technique is employed to simulate the free-surface motion. The second order upwind scheme is used for discretizing the convection terms in the momentum transport equations, the Modified HRIC scheme for VOF discretization. The results obtained compare well with the experimental data.

Keywords**—**Free surface flows, Breaking waves, Boundary layer, Wigley hull, Volume of fluid.

NOMENCLATURE

C	Chord
Re	Reynolds Number
CF	Frictional resistance coefficient Cf = RF/0.5 ρ SV2
S	Source term
Cp	Pressure coefficient Cp = (p / p1)/0.5 ρ V2
V	Velocity vector or speed of advance
CT	Total resistance coefficient CT =RT/0.5 ρ SV2
αa	Volume fraction of air
Fr	Froude number
αw	Volume fraction of water
g	Gravity
ρa	Density of air
k	Turbulent kinetic energy
ρw	Density of water
p	Pressure
ε	Dissipation
Pk	Production
μt	Turbulent viscosity

Saadia Adjali, Mohammed Aounallah, and Mustapha Belkadi are with the Laboratory of Aero-Hydrodynamics Naval, USTO-MB University, P.O. 1505 El-Mnaouar Oran, Algeria (e-mail: adjali.gm25@gmail.com, aounallah_2000@yahoo.fr, belkadigma@yahoo.fr).
Omar Imine is with the Laboratory of Aeronautics and Propulsive Systems, USTO-MB University, P.O. 1505El-Mnaouar Oran, Algeria (e-mail: imine_omar@yahoo.fr).

I. INTRODUCTION

STUDYING the pattern of waves generated by a ship moving through the water is one of the most important objectives in ship hydrodynamics, due to its importance in the design process. The waves produced by a ship in motion can radiate at great distances away from the ship. Furthermore, these waves contain energy that must be dissipated to the surrounding fluid. The ship experiences an opposing force to its movement, one of its components is known as the wavemaking resistance, this being one of the most important components of the ship resistance.

The resistance of a hull is a consequence of air and water forces acting against movement of the vessel. For this reason its determination is an important issue regarding the propulsion and ways to provide it. The water will have the major contribution to the resistance of a hull, unless there are strong winds. As a result, predictions on resistance are a good way to know how the energy is spent. About 30 years ago experimental investigations on a Wigley hull were systematized by ITTC [1], which has been used for validation of numerical methods, including different types of Computational Fluid Dynamics (CFD) codes. During last 15 years more than 30 papers devoted numerical modeling of the Wigley hull instill water and in the waves were published. Some of the papers are based on the assumption that the fluid is incompressible and inviscid and the flow irotational, while other papers use for modeling Reynolds-Averaged Navier-Stokes equations (RANS). Commercial codes (ICCM Comet, Fluent, ANSYS-CFX, STAR CCM, and SHIPFLOW), open code Open FOAM, and several in-house codes were used for simulation. In order to obtain accurate results even in steady state simulations, the problem needs to be set-up carefully and this includes having sufficient nodes within the boundary layer, correct mesh for high gradient zones and suitable time step sizes. Comprehensive efforts have been made to the procedures to verify and validate computational data however there is still a lack of consensus of suitable techniques.

This paper is concerned with the wave generation due to the flow around submerged NACA 0012 moving near the free surface on 2D and Wigleyhull. In early studies of this problem, thin-foil approximation and Neumann type boundary condition were generally used. Hough and Moran [2] and Plotkin [3] used the thin-foil approximation with linearized free surface condition. The former study examined the flow around flat-plate and cambered-arc hydrofoils while the latter included a thickness correction around the leading edge. The Duncan [4] experiment has been the subject of many consecutive studies. Duncan towed a fully submerged 2DNaca0012 profile at a 5° angle of attack, nose up, in steady horizontal motion. At Fr=0.567 (i.e. the wave length λ is equal to twice the chord length c), when the depth d dropped below the chord length c, the first wave in the train began to break. To enhance the short range of transition in terms of the submerged depth from a breaking to a non-breaking wave train, Duncan [4] placed a cloth ahead of the foil and towed it at foil speed for a few seconds before removing it. At critical depth disturbing the free surface resulted in a steady breaking wave train instead of the non-breaking wave train otherwise and the breaking and non-breaking wave resistance. Hino [5] introduced the finite-volume method with an unstructured grid for free surface flow simulation which was based on Euler equations. Kouh et al. [6] analyzed performance of 2D hydrofoil under free surface. They distributed source on undisturbed free surface and doubleton foil and wake surface. Dirichlet-typebody boundary condition is used instead of Neumann-typeboundary condition, the free surface condition is linearized by free stream potential.

In this study, the main focus is laid on the free surface wave generation for submerged NACA 0012 hydrofoil and Wigley hull using VOF method which is a robust, free-surface modeling technique and takes the effects of air into consideration. The interface capturing method is carried out to simulate the problem where both fluids (air and water) are treated as single effective fluid. The method is first applied to NACA 0012 hydrofoil for comparing the results with experimental results of Duncan [4], .in second is then applied to Wigley hull for Fr =0.267 to obtain the wave elevations and total resistance coefficient.

II. Theoretical Formulation

The incompressible viscous flow field around submerged hydrofoil and Wigley hull is simulated with Reynolds Averaged Navier–Stokes (RANS) equation. Using the finite volume method, the governing equation of the flow field and mathematical expression of turbulence model are described as:

$$\rho\frac{Dv_i}{Dt} = -\frac{\partial p}{\partial x_i} + \mu\nabla^2 v_i + \rho g_i \quad \text{i=1, 2, 3} \tag{1}$$

$$\frac{D\rho}{Dt} + \frac{\partial v_i}{\partial x_i} = 0 \tag{2}$$

The treatment for the free-surface flow uses an interface capturing method with the Volume of Fluid (VOF).The VOF method originally developed by [7] is used to compute the surface wave caused by the submerged hydrofoil when moving close to the free surface of water. This model is a fixed grid technique designed for two or more immiscible fluids where the position of the interface between the fluids is part of the unknown to be found through the solution procedure.

In the VOF model, the fluids share a single set of momentum equations, and the volume fraction of each of the fluids in each computational cell is tracked throughout domain. If the αw liquid's volume fraction in the cell noted as αw, then the following three conditions are possible:

- αw＝0, the cell is empty (of liquid)
- αw＝1, the cell is full (of liquid)
- 0<αw<1, the cell contains liquid interface.

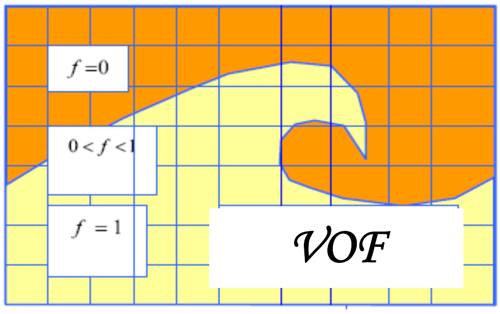

Fig. 1 Advection of fluid

In VOF method, αw is also used to determine the location of interface. The normal direction of the interface lies in the direction in which the value of αw changes most rapidly. The tracking of the interface is accomplished by solving the continuity equation of the volume fraction. For the coefficient αw liquid, this equation has the form:

$$\frac{\partial \alpha w}{\partial t} + \overleftarrow{U}\,\nabla \alpha w = 0 \tag{3}$$

The volume fraction equation will not be solved for air; the volume fraction of air will be computed based on the following constraint:

$$\alpha w + \alpha a = 1 \tag{4}$$

The properties appearing in the transport equations are determined by the presence of the component phases in each control volume. For example, the density in each cell is given by:

$$\rho = \rho w \alpha w + \rho a \alpha a, \tag{5}$$

The viscosity was also computed in a similar manner.

A single momentum equation is solved throughout the domain and the resulting velocity is shared among the phases. The momentum equation is dependent on the volume fractions of all phases through the properties, μ and ρ.

The Shear-Stress Transport (SST) k-ω turbulence model is adopted to calculate eddy viscosity in our study. This model is believed to be one of the best choices to simulate turbulence flow around ship hull. The details of the model formulation can be seen in [8].

$$\frac{\partial k}{\partial t}+\left(U_j\right)\frac{\partial k}{\partial x_j}=P_K-\beta^{*}k\omega+\frac{\partial}{\partial x_j}\left[\left(\nu+\frac{\nu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+-\varepsilon \tag{6}$$

and

$$\frac{\partial \omega}{\partial t}+\left(U_j\right)\frac{\partial \omega}{\partial x_j}=\alpha\frac{\omega}{k}P_K-\beta\omega^2+\frac{\partial}{\partial x_j}\left[\left(\nu+\frac{\nu_t}{\sigma_\omega}\right)\frac{\partial \omega}{\partial x_j}\right]+2\frac{(1-F_1)}{\sigma_{\omega 2}}\frac{1}{\omega}\frac{\partial k}{\partial x_j}\frac{\partial \omega}{\partial x_j} \tag{7}$$

III. Numerical Simulation

In recent years, computation of viscous flow with free-surface effect becomes one of the important subjects in the field of ship hydrodynamics. It has been a challenging research task for years. The presence of free surface makes it more difficult than other flow calculation because the shape and location of free surface is unknown in prior and it's a part of the solution. Using VOF algorithm, the free surface flow around submerged NACA 0012 and Wigley hull is carried out.

At first computational models are created and simulations are performed with NACA 0012 hydrofoil. To validate the computational models for different meshes, computed results are compared with experiment results of [4]. Then the same computational models and simulations are carried out with Wigley hull for Fr=0.267 to observe the free surface water wave caused by the flow around the hull.

A. First Case: NACA 0012 with Free Surface

To construct the computational domain, Gambit (Version 2.3.16) software is used. The geometry of the hydrofoil is created by using standard NACA 0012 coordinates. To mesh the two dimensional domain, it was divided into several regions in order to control the free surface and the hydrofoil boundary layer and wake areas where structured meshes are used, whilst the rest of the domain is meshed using triangles shown in Fig. 2.

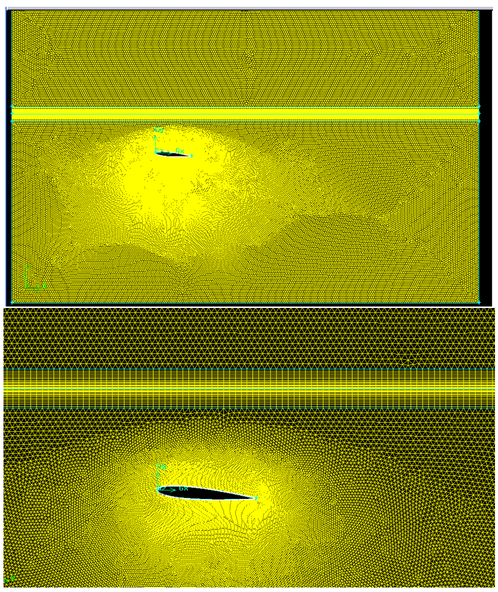

Fig. 2 Mesh detail showing the hydrofoil and the free surface area

The different boundary conditions are used the 'stationary wall' boundary condition is applied for hydrofoil, whereas the 'pressure inlet' and 'pressure outlet' boundary conditions are used for inlet and outlet boundaries respectively and symmetry boundary condition is applied for the upper and lower boundary surface.

B. Results and Discussion

A hydrofoil having chord length 20.3 cm, speed 0.8 m/s, Froude number equal to 0.57, Re = 1592105 and angle of attack 5° is modelled to make validation of the computational results, the simulation of the NACA 0012 hydrofoil is done in the same conditions as the experiment reported by [4]. To check the grid independency of the results, three grids namely Grid 1, Grid 2 and Grid 3 are used in this study. The Grid 1 consists of 68846 cells, Grid 2 has 120644 cells and Grid 3 has 238332 cells. The wave profiles using those three meshes as shown in Fig. 3 refined mesh could produce better results, for our study Grid 3 is used.

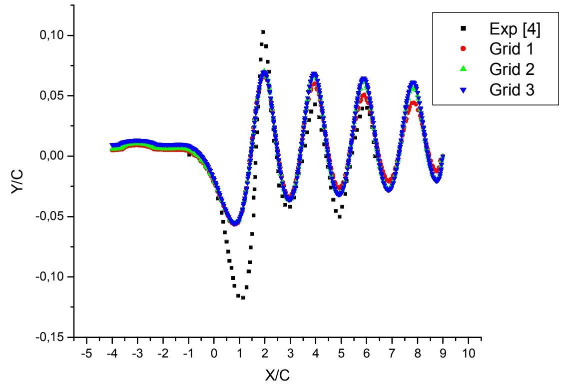

Fig. 3 Comparison between present computational results and experimental results for different mesh

It is observed that the computed wave elevations agree well with experimental wave elevations [4].

The numerical simulation is then carried out. Fig. 4 shows the static pressure around NACA 0012 hydrofoil for depth equal to 0.21 cm. The static pressure increases from the free surface of water (as indicated by the blue color) as the depth increases. The maximum pressure is at the bottom boundary of the domain as indicated by red color.

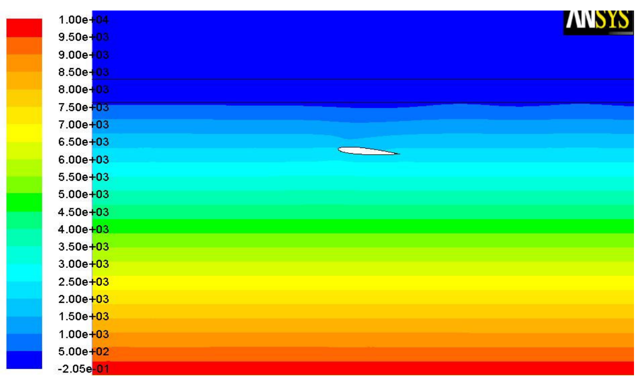

Fig. 4 Contour of static pressure around NACA 0012 hydrofoil for Fr=0.57

Fig. 5 shows the dynamic pressure around NACA 0012 hydrofoil at immersion equal to 21 cm. In this contour, the dynamic pressure at the leading edge and trailing edge of the hydrofoil is lower than the rest of the surface of hydrofoil.

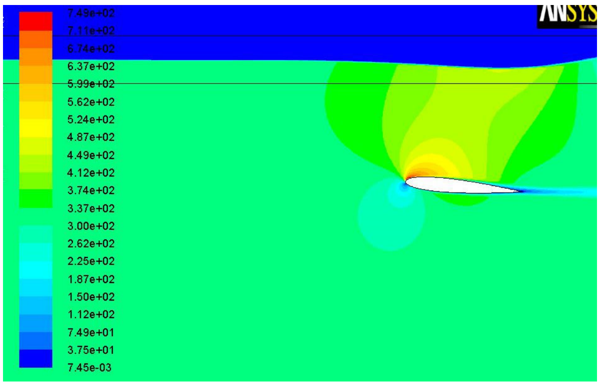

Fig. 5 Contour of dynamic pressure around NACA 0012 hydrofoil at h=21 cm, Fr=0.57

C. Second Case: Wigley Hull with Free Surface

The Wigley hull is a mathematical representation of an actual ship hull and is defined by:

$$Y = 0.5.B(1 - 4\frac{X^2}{L^2})(1 - \frac{Z^2}{T^2}) \qquad (8)$$

It has been widely used in both experimental and numerical studies due to its simple geometry and exact definition. The main dimensions of the hull used are: B /L=0.1, T /L=0.0625. In equation X, Y and Z are the longitudinal, transverse and vertical ordinates of the hull surface and L, B and T are the length, the breadth and the draught of the hull, respectively. For this case L was chosen as 3m.

A physical domain with water and air at standard conditions was specified and a homogeneous coupled Volume of Fluid model was selected as it is recommended for free surface flows where the free surface is well defined over the entire domain. A homogeneous model allows two different phases when the interface is distinct and well defined everywhere. For this initial simulation the k-w SST turbulence model was used.

The total 891396 cells are used for the computational domain; the boundary conditions were imposed as the hull is positioned at a distance 1l after the inlet boundary, 2L before the outlet boundary and 2L in an anther extremity. Meshing of volumes is done by structured elements throughout the domain. The domain size has been proven to be large enough to attain domain-independent solutions; the initial location of the free surface was imposed by defining the volume fraction functions of water and air at the inlet. The boundary conditions were imposed as shown in Fig. 6.

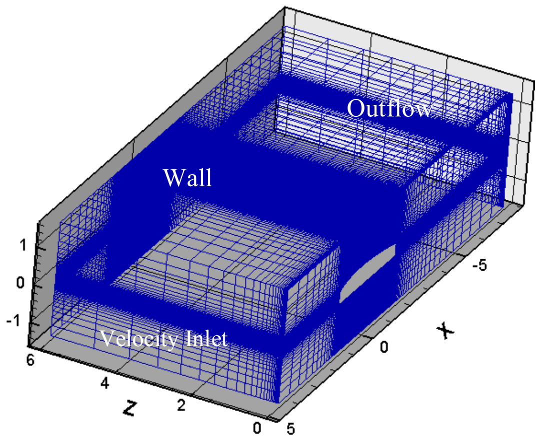

Fig. 6 Initial Computational Domain

D. Results and Discussion

The simulated wave-elevation contours for Fr=0.267 is shown in Fig. 7:

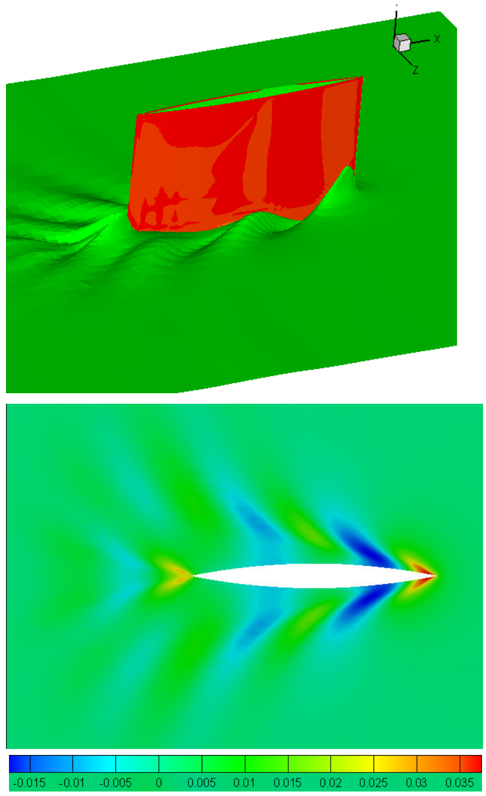

Fig. 7 Wave contours for Wigley hull, Fr=0.267

For a more quantitative analysis, the wave elevation profile obtained is compared against with experimental data as shown in the graph below. In such a graph, wave elevation is normalized to the length of the Wigley hull.

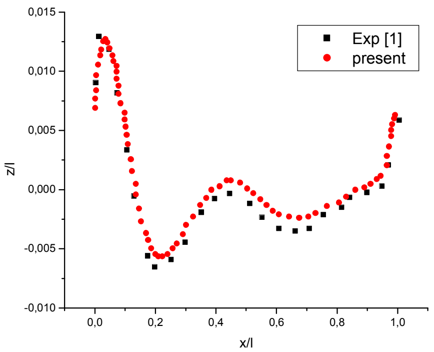

Fig. 8 Wave profile along wigley hull, Fr=0.267

The comparisons between the predicted results and the available experimental results [1] show in general good agreement between the results at Fr = 0.267. It can be seen that the wave profile near the bow is a trough of 0.0125L after crest of -0.007L

Traditionally, the result of model resistance test can be expressed as a corresponding relation between resistance coefficient and Froude number in a dimensionless way. The total resistance coefficient is generally divided into two parts, frictional and residuary resistance coefficient. The frictional resistance coefficient is obviously a function of Reynolds number for viscous effect, whereas the residuary resistance coefficient almost has no correlation with Reynolds number. The numerical results obtained for the friction resistance are presented in Table I by Fluent.

TABLE I
NUMERICAL RESISTANCE COMPONENTS OF THE WIGLEY MODEL FR =0.267

Fr	C_F	$C_{F(ITTC57)}$	C_W	C_T
0.267	$3.1.10^{-3}$	$3.48.10^{-3}$	$0.843.10^{-3}$	$3.943.10^{-3}$

The comparisons between the predicted results and the values obtained by ITTC method show in general good agreement between the results at Fr=0.267. Friction coefficient predicted by fluent is somewhat great predicted than that obtained from ITTC method, this difference is likely to have been caused by rate of refinement of mesh. The comparisons between the experimental results and Fluent results in the previous table show the ability of the RANSE code CFX in predicting accurate values for the resistance coefficients.

The detected distribution of the pressure coefficient on the Wigley hull at Fr = 0.267 is presented in Fig. 9. The zones of high pressure at the bow and the aft regions are detected well by the code. The higher value of the pressure coefficient can be found at the stagnation region on the hull. There is a pressure gradient aft of the stagnation pressure region, the pressure increases again; finally the hull pressure distribution was detected well by the code.

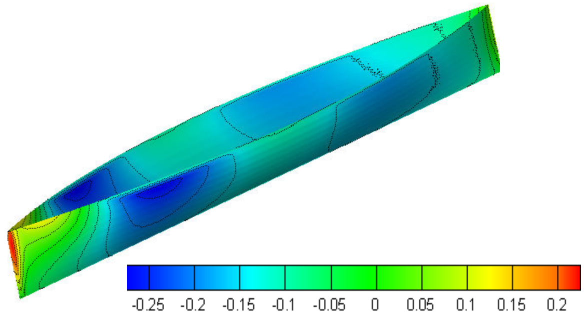

Fig. 9 Distribution of Cp on the hull surface of Wigley model at Fr = 0.267

IV. CONCLUSION

The turbulent free surface flow around NACA 0012 and Wigley hull has been simulated using the RANSE code Fluent. The wave profiles along the submerged NACA 0012 and Wigley hull well against the available experimental results. The results of Fluent code show the ability of predicting good results for the free surface wave pattern for the both cases, even with the need for more computational grid.

The most important conclusions that can be drawn from this study are:

- Two-dimensional implicit finite volume method is successful for the analysis of flow around hydrofoil.
- The volume of fluid (VOF) method along with realizable k–ω SST turbulence model can satisfactorily predict wave generated by the flow around hydrofoil moving near free surface. And for predicting the frictional resistance and the viscous pressure components of Wigley hull at Fr=0.267
- The good benefit from using the hybrid mesh with a RANSE solver and limited computational resources for predicting the turbulent free surface flow pattern around a submerged NACA 0012 without the use of very large number of computational grid elements. The present method also computes hydrodynamic forces satisfactorily.

REFERENCES

[1] ITTC "Cooperative Experiments on Wigely Parabolic Models", (17th ITTC Resistance Committee Report, 2nd Ed, Japan, 1983).
[2] Hough, G. R, "Moran, and S. P: Froude number effects on two-dimensional hydrofoils, J. Ship Res. 13, 53–60, 1969.
[3] Plotkin, A., Thin-hydrofoil thickness problem including leading-edge corrections", J. Ship Res. 19, 122–129, 1975.
[4] Duncan, J. H. "The breaking and non-breaking wave resistance of a two dimensional hydrofoil", J. Fluid Mech. 126, 1983.
[5] Hino, T, "A finite-volume method with unstructured grid for free surface flow simulations", Proceedings of the 6th International Conference on Numerical Ship Hydro, Iwoa, USA, 1993.
[6] Kouh, J.S., Lin, T.J., Chau, S.W," Performance analysis of two-dimensional hydrofoil under free surface. ", J. Natl. Taiwan Univ, 86, 2002.
[7] Hirt, C. W., Nichols, B. D, "Volume of fluid (VOF) method for the dynamics of free boundaries", J. Comput. Phys. 39 (1), 201–225, 1981.
[8] Fluent Inc, User Guide, 2012.