Two-phase Flow Analysis for Small-scale Ballast Water Tank Model by Hydraulics Experiment and Simulations

Guangshuai Liu¹, Toraharu Watanabe² and Takeshi Shinoda³

¹ Department of Urban and Environmental, Graduate School of Engineering, Kyushu University, Japan
² Department of Naval Architecture and Ocean, Graduate School of Engineering, Kyushu University, Japan
³ Department of Marine Systems Engineering, Faculty of Engineering, Kyushu University, Japan

liu.guangshuai.228@s.kyushu-u.ac.jp

Abstract. This study is utilizing hydraulics experimental and computational fluid dynamics (CFD) methods to verify the flow phenomenon in the ballast water drainage process and discuss the drainage efficiency. In order to facilitate the analysis of the flow characteristics of ballast water in the ballast tank of a typical bulk carrier, the complex structure is simplified into two simple small-scale models, a single-longitudinal model and a single-transversal model. Both are designed with the scale of 1:10 for the data collection. The experimental results are then validated and verified by conducting a two-phase flow model applied volume of fluid (VOF) method. Based on that result, the use of the VOF method is suitable for the ballast water drainage simulation, so it can be used in large-scale model calculations to predict the drainage efficiency of ballast tanks. In addition, this study also conducts a comparative analysis of two configuration modes of the single-longitudinal model and the single-transversal model, which provides a reference for the design and improvement of complex structures.

1. Introduction
Ships, especially those carrying large quantities of specific cargo, inevitably face the situation of sailing without cargo. In order to obtain sufficient seaworthiness conditions in terms of stability, trim, strength, and propulsion efficiency, seawater is pumped into the ballast tank to form the ballast condition. Moreover, for some special loading conditions, ballast water is also responsible for adjusting the ship’s navigation attitude and stability compensation.

At present, researches on ballast water in the world mainly focus on species invasion and environmental pollution caused by ballast water exchange [1] but the drainage efficiency is rarely mentioned. The drainage of ballast water is directly related to the added value of shipping companies. This mainly involves two aspects: 1. Drainage time. If the drainage time is shortened, the time for berthing in the harbour can be reduced, which also saves the rental cost of the port berth; 2. Residual water volume. If the amount of residual water is reduced as much as possible, the deadweight of the ship can be more fully utilized, which increases the shipping profit.
However, most ballast tanks are structurally complex as shown in Figure 1, which are composed of compartments separated by girders, longitudinals, transverses, etc., which are interconnected by lightening holes and drain holes, etc. In the design of the ballast water system, the configuration of drain holes, pump and bell-mouths, and the choice of drainage methods are related to structural strength and construction costs. Up to now, there is no design method that can solve the above two problems related to the efficiency of ballast water drainage as far as possible.

The objective of this study is to develop an experimentally validated computational model that can be used to study flow phenomena in ballast tanks and to evaluate the drainage efficiency, so as to provide references for ballast tank design optimization. The present work starts from the analysis of simple models. Taking the ballast tank of a typical bulk carrier as the research object, small-scale models of two arrangements with a scale of 1:10 are made to collect experimental data. Then the numerical simulation is carried out with the computational fluid dynamics method, and compare with the experimental results, so as to verify the feasibility of numerical calculation for ballast water drainage simulation. The analysis of two arrangements can also help to understand the characteristics of ballast water flow, and provide experience for simulation analysis and model mesh optimization of large-scale models. It might also provide a reference for the improvement of ballast tank design.

2. Model design
In this paper, the area enclosed by the transverse and longitudinal in the double bottom framing system is called a compartment. The area enclosed by sider girders and transverses which is composed of multiple adjacent compartments divided by longitudinal is called a large compartment. Define the positive direction of the X-axis from the stern to the bow, and the positive direction of the Y-axis from the centerline to the port side. Therefore, the arrangement of the compartments can be expressed as $M \times N$ ($M$ is the amount in the $X$ direction, $N$ is the amount in the $Y$ direction). And the location of a compartment can be expressed as $(m-n)$ ($m$ is the position in the $X$ direction, $n$ is the position in the $Y$ direction).

As shown in Figure 2, the hydraulic experimental equipment in this study consists of a ballast tank model, a ballast pump system, and an ultrasonic sensor. The 1/10 scale model is designed according to the ballast tank of a typical bulk carrier, compartments are arranged as $8 \times 8$ with opening tops. It is made of transparent acrylic material, and the thickness of the frame plate is 2 mm. Longitudinal and transverse can be freely detached and combined by tenons, and drain holes of different sizes can be replaced. This also allows experiments with fewer compartments than $8 \times 8$ to be completed. The size of a compartment is $260 \text{ mm} \times 77 \text{ mm}$ (length in the $X$ direction and length in the $Y$ direction), the height of the longitudinal is 30 mm, and the height of the transverse is 80 mm. It should be noted that "1:10 scale" refers to the scaling in geometric length. The length and width are 1/10 of the full-scale tank, and the height of the longitudinal is also 1/10 of the full size. Since the transverse bone is usually a closed structure in a double bottom, only a suitable height is selected. The design of the experimental
model does not introduce the consideration of non-dimensional parameters, but this can be an extension in future research. In addition, in order to facilitate model making and numerical simulation, the design omits fine structures, such as stiffeners and panels of longitudinal, and simplifies circular holes to square. The influence of these structural changes will also be explored in future research.

In the M × N-arranged ballast tanks, for each compartment, there are multiple incoming flows from adjacent compartments, which is difficult to analyze the flow characteristics. Therefore, simplifying the model to a single-longitudinal or single-transversal arrangement is significant for observing and analyzing flow characteristics.

Figure 2. Experimental equipment.

Figure 3. Small-scale ballast water tank models.

Table 1. Model and experiment settings

| Case | Tank Model No. | Longi. Height (mm) | Trans. Height (mm) | Drain Hole B.×H. (mm) | Pumping Rate (L/min.) | Initial Waterlevel (mm) | Bell-mouth Position |
|------|----------------|--------------------|-------------------|-----------------------|-----------------------|------------------------|------------------|
| Case 1 | Tank model 1 | - | 80 | 9x6 | 3.3 | 30 | (1-1) |
| Case 2 | Tank model 2 | 30 | - | 9x6 | 3.3 | 30 | (1-1) |
| Case 3 | Tank model 2 | 30 | - | 9x6 | 3.3 | 50 | (1-1) |
| Case 4 | Tank model 3 | 30, 80⁴ | - | 9x6 | 3.3 | 50 | (1-1) |

⁴ Only the height of Longi. 3 is 80mm, the rest are 30mm.
In four cases, three models shown in Figure 3 are utilized and their characteristics are given in Table 1. In Case 1, Tank model 1 is a single-longitudinal model (6×1). Since the transverse of the typical bulk carrier is a closed structure and the longitudinal heights are 30 mm, there is only flow through the drain holes in the adjacent compartments in Tank model 1 (Figure 3(a)), so the initial water level is set as 30 mm which is sufficient. Case 2 and Case 3 use the same model (Figure 3(b)), and the height of longitudinal is set to 30 mm. Since there is the flow above the longitudinal while the water level is higher than 30 mm, the initial water level is analyzed according to two conditions. Due to the structural characteristics of the ballast tanks in the width direction of the ship, Case 4 replaced the third longitudinal (Longi. 3) with a sided girder (Figure 3(c)) to observe the flow phenomenon between large compartments. In this case, the initial water level at 30 mm is the same as the situation in Case 2, so only the initial water level at 50mm is simulated. The bell-mouth of each case is located in the center of compartment (1-1). The cases in this paper are all in horizontal conditions, although the trim state will also affect the drainage of the ballast tank, which will be studied in future cases.

A diaphragm pump (TACMINA, FXD-FXW-8) is used as a ballast pump system, which is composed of three reciprocating diaphragm pumps with different phases to reduce pump pulsation, so that the suction can be stable. Also, for the convenience of model making and numerical simulation, bell-mouth is simplified as a long straight pipe with an outer diameter of 32 mm and an inner diameter of 25 mm, made of vinyl chloride. It is inserted from the top of the tank and placed with the lower edge at a height of 4 mm from the bottom.

Ultrasonic sensors (KEYENCE, FW-02) are used to measure the change over time in the water level, and distance measurement is based on the principle of ultrasonic reflection. The measuring range is 50–200 mm, and the response speed is 250 milliseconds.

3. Experimental approach

The output of the pump can be changed by the frequency with an inverter to adjust the flow rate. In the experiment, the flow rate of the pump is set to 3.3 L/min, which is equivalent to 200 ton/h of a full-scale tank. This value is less than normal working conditions which is 1000 ton/h, because the high speed may cause difficulty in the convergence of the results in the numerical calculation. And the freshwater (tap water) is used to instead seawater or saltwater.

The output data of the ultrasonic sensor is a voltage signal rather than a direct distance and has a linear relationship with the distance. So, the experimental water level can be obtained according to the following method. First, inject water into the ballast tank model to reach the initial water level \( w_i \), and let it stand. The state of the water surface can be observed by the display screen of the ultrasonic sensor. When the voltage signal becomes stable, the standing of water can be considered to be completed. Before switching on the pump, turn on the data logger (time \( t_0 \)) to record initial data for 5 seconds or more. Then start the pump (time \( t_1 \)) and observe the change of the water level in the tank. When the water level drops to almost stable (time \( t_2 \)), record the final water level \( w_f \), then turn off the pump and wait about 30 seconds before stopping the data logger (time \( t_3 \)) to store enough voltage data corresponding to the final water level. Finally, the water level movement curve of the experiment is obtained according to equation (1).

\[
w(t) = w_i - \frac{w_i - w_f}{\bar{V}_f - \bar{V}_i} \left( V(t) - \bar{V}_i \right) + w_i
\]

where \( V(t) \) and \( w(t) \) are the voltage and water level at time \( t \). \( \bar{V}_i \) is the initial average voltage in time period \( t_0 \sim t_1 \) and \( \bar{V}_f \) is the final average voltage in time period \( t_2 \sim t_3 \).

It should be noted that the use of this measurement method has drawbacks. Ultrasonic sensors can only measure the water level change of a point, not a larger area. If the water surface has a large curvature, the water level characteristics of the area are difficult to be expressed. Therefore, two measurement locations are set in the compartment where the bell-mouth located. Moreover, since it is only a point of the water level, the measurement results will fluctuate greatly.
4. Computational Method
In this research, the open-source CFD software OpenFOAM (version 5.0) is used for CFD simulation. The advantage of this software is supporting the addition and modification of privately developed code, which gives more flexibility in controlling the algorithms.

OpenFOAM uses a mesh-based finite volume method (FVM) to solve the governing equations of mass and momentum. Solve the transient flow of incompressible fluids. The turbulence model is the Reynolds-Averaged Navier-Stokes (RANS) standard k-ε model. It should be noted that the RANS model is called the RAS (Reynolds-Averaged Simulation) model in OpenFOAM. During the discharge of ballast water, there are two-phase fluids of air and water in the tank, and the volume of fluid (VOF) [2] method is utilized for tracking the water surface. This method is widely used in the fields of ship navigation [3], aerospace [4], materials [5], metallurgy [6], printing technology [7], etc.

4.1. Volume of fluid method
In the VOF method, an indicator function is defined to represent the phase fraction of the fluid. For insoluble and incompressible multiphase systems, the transport equation for the indicator function, representing one phase is solved simultaneously with the continuity and momentum equations.

\[
\nabla \cdot \mathbf{U} = 0 \tag{2}
\]

\[
\frac{\partial \rho \mathbf{U}}{\partial t} + (\mathbf{U} \cdot \nabla) \rho \mathbf{U} = -\nabla p + \mu \nabla^2 \mathbf{U} + \mathbf{F} \tag{3}
\]

\[
\frac{\partial \alpha}{\partial t} + \nabla \cdot (\alpha \mathbf{U}) = 0 \tag{4}
\]

where \( \mathbf{U} \) is the flow velocity, \( \rho \) is the fluid density, \( p \) is the pressure, \( \mu \) is the dynamic viscosity, \( \mathbf{F} \) is the body forces. Consider the gas-liquid two-phase system, the indicator function \( \alpha \) is commonly defined in each computational cell as: if the cell is filled with liquid, then \( \alpha = 1 \); if the cell is filled with gas, then \( \alpha = 0 \); if there is a gas-liquid mixing condition, \( \alpha \) is between 0 and 1. It can be seen that the interface in the VOF method is obtained by calculating the phase-field value in each cell and then post-processing it, not by directly calculating the interface. The advantage of this method is that the volume fraction is introduced to implicitly represent the phase interface, and no complicated phase interface tracking algorithm is needed. It is very important for the calculation of the two-phase flow modeled by complex geometric shapes.

The critical issue in numerical simulations of free-surface flows using the VOF model is the sharp resolution of the interface while preserving the boundedness and conservation of the phase fraction [8]. It is well known that the numerical discretization of equation (4) will produce numerical diffusion at the interface. Although it is feasible to use a very fine mesh is used to reduce the error in phase fraction, it will result in higher requirements for computing resources. Therefore, remedies need to be introduced to sharpen the interface and many methods for sharpening the interface [3][4][8][9] have been attempted.

4.2. Interface sharpening method
The basic solver that uses the VOF method to solve multiphase flow problems is interFoam [10]. In order to ensure that the interface is sharp enough, the method proposed by Weller [11] is utilized. This method compresses the phase fraction near the interface by introducing an artificial compression with in the phase fraction equation (4) to counteract the smearing of the interface caused by numerical diffusion. In addition, the value of the artificial convection term must be zero where it is not the interface. According to this idea, the phase equation can be written as:

\[
\frac{\partial \alpha}{\partial t} + \nabla \cdot (\alpha \mathbf{U}) + \nabla \cdot (\alpha(1 - \alpha) \mathbf{U}_c) = 0 \tag{5}
\]
The last term on the left-hand side of equation (5) is non-zero only at the interface due to \( \alpha(1 - \alpha) \). \( U_\alpha \) is used to ensure the compression of the interface. It must ensure that the compression term will not bias the solution in any way, so its direction can only be the same as the normal direction of the interface. And, it should not exceed the maximum velocity near the interface. Weller [11] suggests the compression velocity can be calculated as:

\[
U_\alpha = c_\alpha |U| \frac{\nabla \alpha}{|\nabla \alpha|}
\]

(6)

The intensity of the interface compression is controlled by the compression factor \( c_\alpha \). Zero means no compression; 1 means conservative compression; greater than 1 means enhanced compression [10]. In this work, \( c_\alpha \) is set as 1.

4.3. Time step control
The time increments are controlled according to the Courant–Friedrichs–Lewy (CFL) condition in the form

\[
\Delta t < \min \left( \frac{dx}{u}, \frac{dy}{v}, \frac{dz}{w} \right)
\]

(7)

where this minimum is with respect to every cell in the grid, not just at the interface. \( dx, dy, dz \) are the cell size in three directions, \( u, v, w \) are the components of the cell's velocity in three directions. The functions of automatic time step control [9] is provided, and the time step is forced to be adjusted to ensure that the data is saved at the exact times specified for write output.

4.4. Mesh
In OpenFOAM, a hexahedral unstructured mesh is used in this study. The difficulty in constructing the hexahedral mesh lies in the problem of fitting the mesh to the curved surface when there is a space, where the transition from a square structure to a spherical structure occurs, in the computational domain. The built-in meshing tool is not easy to operate, so another meshing software ICEM is utilized to process the mesh near the surface. As shown in Figure 4, the Ogrid-Block tool is utilized so that the hexahedral mesh can fit the curved surface well. Thereby the area or volume loss of obstacles is reduced in the computational domain.

Figure 4. The mesh distribution around the cylindrical structure (cross-section).

Figure 5. The mesh distribution in the height direction.

Figure 6. The mesh distribution in the horizontal direction.
Table 2. Model arrangements and meshes

| Case   | Software | Arrangement | Cells   |
|--------|----------|-------------|---------|
| Case 1 | OpenFOAM | 6x1         | 147,016 |
| Case 2 | OpenFOAM | 1x6         | 165,594 |
| Case 3 | OpenFOAM | 1x6         | 165,594 |
| Case 4 | OpenFOAM | 1x6         | 164,874 |

As shown in Figure 5 and 6, a cut plane at the pump bell-mouth centerline and another horizontal cut plane near the bottom of the tank show the structure of the mesh. The mesh is concentrated in the region of interest, such as near the bell-mouth and through the drain holes near the bottom of the tank, to improve resolution. The total number of cells for each case is given in Table 2. The methods of mesh distribution in the four cases are approximately the same, and the number of cells varies due to the variation of the structural configuration.

4.5. Post-processing
In order to facilitate the comparison between the simulation and the experiment, data is extracted and processed from result files by programming. The iso-surface with the phase fraction $\alpha = 0.5$ is taken as the water surface, and the corresponding height of each cell is extracted as the water level. The average water level of each compartment is calculated according to the following formula,

$$
H_{mn} = \frac{\sum h_{ij} \times d_{ij}}{\sum d_{ij}}
$$

where $H_{mn}$ is the average water level of tank m-n, $h_{ij}$ is the height corresponding to $\alpha = 0.5$ of each cell, $d_{ij}$ is the projected area of each cell perpendicular to the direction of water level. And the total volume of discharged water $V_d$, can be calculated as:

$$
V_d = \sum (H_1 - H_{mn}) \times S_{mn}
$$

where $H_1$ is the initial water level. $S_{mn}$ is the bottom area of each tank.

5. Validation Results
Since the actual pumping rate in the experiments has an error with the set value, the pumping rate set in the simulation should be adjusted according to the experimental measurement results. In addition, the physical properties of the fluid and the ambient temperature are also set to approximate the experiment to ensure reliable results.

5.1. The single-longitudinal arrangement
As shown in Table 1, the initial water level of Case 1 is 30 mm. The simulation is carried out until the total elapsed time reached 60.0 seconds, which is enough to simulate the characteristics of water level changes in each compartment and the relationship between adjacent area.

The waterlevel-time curves of the experiment and simulation are shown in Figure 7(a). The water level drop rate in six compartments gradually slows down with further away bell-mouth. Due to the limitations of the measurement method, the water level of the experiment (solid lines) fluctuates greatly in local changes, but the overall trend is clear. For the simulation results of OpenFOAM, since the water level is calculated according to the averaging method in Section 4.5, the curves are smoother. According to the comparison, the OpenFOAM can well simulate the characteristics of water level change ballast tanks during drainage.
5.2. The single-transverse arrangement
The remaining three cases are single-transverse arrangements, and the experiments are carried out according to the settings shown in Table 1. Among them, Case 2 is simulated until 120 seconds, and Case 3 and 4 are simulated until 180 seconds.

The comparisons of the water level and the discharge volume of simulation and experimental results of each case are given in Figure 8, Figure 9 and Figure 10. The simulation results are in good agreement with the experiments. The decreasing trend of the water level in each compartment in Case 2 is similar to that in Case 1. In Case 3, the water level drop rate in all compartments is similar in the first 30 seconds. When the overall water level drops to where close to the upper edge of the longitudinal, a point of divergence occurs at this time. During the period from this point down to the upper edge of the longitudinal, there is a waterfall-like height difference between adjacent compartments shown in Figure 13(b). The trend of the water level below the longitudinal height is similar to Case 2. In Case 4, the water level of six compartments is divided into two groups, and drop at different rates: (1-1), (1-2), and (1-3) compartments remain similar until about 20 seconds; (1-4), (1-5) and (1-6) compartments remain similar until about 70 seconds. When the water surface approaches the upper edge of the longitudinal, divergence points occur similar to that in Case 3. After that, the falling trend of the water level in each compartment is also similar to Case 2.
Figure 9. Result comparison for experiment and OpenFOAM of Case 3.

Figure 10. Result comparison for experiment and OpenFOAM of Case 4.

Figure 11. Velocity distribution between bell-mouth and drain hole.

In Case 3, a cut plane at the bell-mouth centerline and bisects the drain hole showing the water velocity distribution at 20 seconds, 32 seconds and 40 seconds are given in Figure 11. When the water level is much higher than the upper edge of the longitudinal in Figure 11(a), the flow between adjacent compartments mainly exists in the space above the longitudinal, which at a lower speed. When the water level approaches the upper edge of the longitudinal in Figure 11(b), the velocity in the drain hole begins to increase and be affected by the suction force of the pump. When the water level is lower than the height of the longitudinal in Figure 11(c), the flow between compartments is completely inclined to through the drain hole, and the velocity in the drain hole is affected by the pump to the maximum.

Based on the above comparisons, the credibility of CFD simulation can be verified. The relationship between water level changes and ballast tank frame layout can also be summarized: when the water level is much higher than the upper edge of the longitudinal, the adjacent compartments can
be regarded as a whole; with the water level is falling close to the height of the longitudinal, a divergence point occurs; after the water surface is lower than the height of the longitudinal, the water level change is only related to the flow in drain holes. In addition, the divergence point is slightly higher than the upper edge of the longitudinal, and the regularity needs further research to conclude.

6. **Comparison of longitudinal and transverse arrangement**

As shown in Case 1 in Figure 7(a) and Case 2 in Figure 8(a) that the falling trend of water level under the two arrangements is similar. The discharge volume comparison of the two arrangements in the simulation at the same pumping rate is given in Figure 12. In the first 10 seconds, the discharge volumes of the two cases are similar. After 10 seconds, the discharge volume of Case 2 is higher than that of Case 1, which means that the discharge efficiency of ballast tanks arranged transversely is higher than that of longitudinally arranged tanks.

Horizontal cut planes at a height of 3 mm from the bottom showing the water velocity distribution at 5 and 20 seconds are given in Figure 13, And Case 2 is rotated 90° clockwise for comparison. In the initial stage, the water surface in the compartment where the bell-mouth locates is much higher than the lower edge of the bell-mouth. At this time, the water discharged mainly comes from the bell-mouth compartment, and the suction of two pumps has almost the same effect on the flow in the drain hole, so, two arrangements can maintain a similar discharge volume.

![Figure 12. Comparison for different arrangements: Case 1 and Case 2.](image)

![Figure 13. Velocity distribution of different arrangements.](image)
When the water surface is approaching to the lower edge of the bell-mouth, the suction of the pump in Case 2 has a higher impact on the flow of the drain hole than in Case 1, because the Case 2 bell-mouth is closer to the drain hole. Therefore, the suction effect on the flow in the drain hole also promotes the overall drainage efficiency. The magnitude of the pump suction effect on the hole is related to the distance from bell-mouth to the drain hole and the depth of the bell-mouth underwater.

7. Conclusions
Experimental and CFD simulation methods have been utilized to verify the flow behaviour in the ballast tank during the discharge of ballast water and to calculate the discharged volume. The research introduces the idea of splitting the complex ballast tank into two types of simple arrangements in order to find the flow characteristics. The verification work is carried out using the ballast tank size of a typical bulk carrier to make a 1:10 scale model. The results of the verification study show that the results of CFD simulation and experimental measurement are in good agreement. According to the results obtained from several arrangement patterns, the regularity of water level drop affected by the structure during the discharge of ballast water is concluded. And the divergence points of the water level change and the pump suction effect on the flow of drain holes still need further research to verification.

The work in this paper provides a method and theoretical basis for the CFD simulation of the complex ballast tank, and also provides a reference for the improvement and analysis of the structure and equipment arrangement in the ballast system. In addition, a simplified mathematical model of the ballast system will be developed and optimized based on this study, in order to improve a more efficient evaluation program for the design of ballast tanks.

References
[1] Wilson W, Chang P, Verosto S, Atsavapranee P, Reid D F and Jenkins P T 2006 Computational and experimental analysis of ballast water exchange Nav. Eng. J. 118 (3) 25-36
[2] Hirt C W and Nichols B D 1981 Volume of fluid (VOF) method for the dynamics of free boundaries J. Comput. Phys. 39(1) 201-25
[3] Chen S, Zhong J and Sun P 2015 Numerical simulation of the stokes wave for the flow around a ship hull coupled with the VOF model Journal of Marine Science and Application 14 163-9
[4] Bertolotti L, Jefferson Loveday R, Ambrose S and Korsukova E 2021 A comparison of volume of fluid and Euler–Euler approaches in computational fluid dynamics modeling of two-phase flows with a sharp interface J. Turbomach 143(12) 121005(8)
[5] Yang Y, Cai S, Yao J, Zhong J, Zhang K, Song W, Zhang L, Sun Hai and Lisitsa V 2021 Pore-scale simulation of remaining oil distribution in 3D porous media affected by wettability and capillarity based on Volume of Fluid method International Journal of Multiphase Flow 143 103746(14)
[6] Ramirez-Argáez M A, Dutta A, Amaro-Villeda A, González-Rivera C and Conejo A N 2019 A novel multiphase methodology simulating three phase flows in a steel ladle Processes 3 175
[7] Orlova E, Chernaya I V, Sultanova Y, Rovenskikh A A, Varepo L and Trapeznikova O 2021 Practice of the VOF OpenFOAM techniques aimed to parameters prediction of the «dampening solution - paper» interaction for the printing systems Journal of Physics: Conference Series 1791 012104(9)
[8] Giussani F, Montorfano A, Piscaglia F, Onorati A, Hélie J and Aithal S 2016 Dynamic VOF modelling of the internal flow in GDI fuel injectors Energy Procedia 101 574-81
[9] CHAM Ltd. 2019 PHOENICS Manual vol 2 (London: CHAM Ltd.) chapter 10 p 16
[10] Greenshields C J 2017 OpenFOAM User Guide (Delaware: OpenFOAM Foundation Ltd.) pp 60-72
[11] Weller H 2008 A new approach to vof-based interface capturing methods for incompressible and compressible flow OpenCFD Ltd. Technical Report TR/HGW/04