Computational Fluid Dynamics of Dam-Break Problem with Different Fluid Properties Using the Level Set Method

Ashraf Balabel[1]*, Ali Alzaed[2]

[1] Mechanical Engineering Department, Faculty of Engineering, Taif University, Taif, Saudi Arabia
[2] Architectural Engineering Department, Faculty of Engineering, Taif University, Taif, Saudi Arabia

Email Address

ashrafbalabel@yahoo.com (ashraf Balabel)
*Correspondence: ashrafbalabel@yahoo.com

Received: 18 December 2017**; Accepted:** 20 January 2018**; Published:** 30 March 2018

Abstract:

In the present paper, the computational fluid dynamics for one of the most challenging problems of two-phase flows; namely, the unsteady movements of dam break flow, is introduced. The governing momentum equations are solved numerically using the control volume approach over a regular two-dimensional computational grid. The level set method is applied for predicting the transient movements of the dam break free surface flow with a high accuracy solution procedure. The effects of the geometrical parameters of the initial dam shape and the fluid properties such as density, viscosity and surface tension on the dam front movement and dam topological changes are investigated. The obtained results showed a faster movement of the dam front in the downstream direction by increasing the dam height and the fluid density. However, by increasing the fluid viscosity, a slower motion of the dam front is obtained. Moreover, the variation of the fluid surface tension showed a slight effect on the movement of the dam front.

Keywords:

Computational Fluid Dynamics, Dam Break Problem, Fluid Properties, Level Set Method, Two-Phase Flow

1. Introduction

Dam break problem is considered to be one of the most important two-phase applications in engineering and industrial fields, e.g. hydropower stations, marine hydrodynamics and coastal engineering. Consequently, dam break flow has been the subject of extensive experimental, theoretical and numerical investigations for more than a hundred year. Detailed literature review for such problem can be found in [1].

Experimental studies on dam break problem have observed the shape of the water propagation front and its traveling distance in the horizontal direction. In particular, the representative experimental investigation, which is considered as a benchmark test for dam break problem, is referred to [2]. This experiment was later repeated using other experimental techniques by [3, 4].

According to the experimental complexity of such problem, exact measurements of the interface shape are not available; however, some measuring data such as the reduction of the water column height and the horizontal traveling distance of the water front can be applied for the validation of the numerical methods through the quantitative comparison of the obtained results.

Moreover, the theoretical as well as the numerical treatments of the dam break problem have encountered some constraints related to the nature of the problem as it includes a transient, non-uniform free surface flow with large spatial and temporal gradients. Moreover, turbulence characteristics should be considered with the moving wave front which is driven by gravity developed turbulence. Therefore, most of the previous numerical treatments of such problem are usually simulated by neglecting turbulent stresses or using various empirical assumptions about the parameters of turbulence. Recently, there is a trend to replace the previous attempts with direct numerical simulations, large eddy simulations and advance turbulence modeling, see for more details [5, 6].

According to the huge development in the numerical simulation of turbulent two-phase flows with wide range of length scales, carefully executed simulations in such context can virtually replace experiments [7]. In general, the numerical predictions of turbulent dam break dynamics have been limited in accuracy partly by the performance of three key elements, viz.: development of the computational algorithm, interface tracking methods, and turbulence prediction models [8-15].

A variety of numerical methods have been recently developed and validated to two-phase. However, an efficient and complete numerical method is not available. An extended review of numerical techniques applied for turbulent two-phase flow including adv/disadvantages can be found in [16]. More recently, the author has developed a new numerical method, known as Interfacial Marker Level Set Method (IMLS), for predicting the complete dynamics of turbulent two-phase flows. This numerical method is validated and its accuracy is estimated through the performing of a wide range of industrial and engineering applications [17]. This method is also further applied in the present paper and shortly explained in the following sections.

2. Governing Equations

The governing equations for incompressible flow are mathematically expressed by the conservation equations of the mass and momentum at each point of the flow field. These equations can be written in the primitive variables formulation in form of continuity equation and Navier-Stokes equations, respectively, as follows:

$$\nabla \cdot \mathrm{u} = 0 \tag{1}$$

$$\rho\left(\frac{\partial \mathrm{u}}{\partial t} + \nabla \cdot (\mathrm{uu})\right) = \nabla p + \mu \nabla^2 \mathrm{u} \tag{2}$$

where r, u, p, m are the density, velocity vector, pressure and viscosity of the fluid. The fact that there is no pressure transport equation necessitates the consideration of the continuity equation as a means to obtain the correct pressure field. This is done by a proper coupling between the pressure and velocity field through the Poisson equation for pressure:

$$\nabla^2 p = -\nabla \cdot \rho \left(\frac{\partial \mathrm{u}}{\partial t} + \nabla \cdot (\mathrm{uu}) \right) \tag{3}$$

The Poisson equation is solved by means of the Successive Over-Relaxation method.

3. Level Set Method

The level set method is a type of capturing methods where a defined phase function ϕ, is smoothed over the entire computational domain. The level set function at any given point is taken as the signed normal distance from the interface with positive on the liquid phase (i.e. ϕ>0), and negative on the gas phase (i.e. ϕ<0). Consequently, the interface is implicitly defined as the zero level set of the level set function.

The transport equation of the level set function can be described by the following equation:

$$\frac{\partial \phi}{\partial t} + \bar{u} \cdot \nabla \phi = 0 \tag{4}$$

where $\bar{u}$ is the velocity vector. The geometrical properties of the interface, (normal vectors and curvature), can be defined as:

$$\bar{n} = \frac{\nabla \phi}{|\nabla \phi|}, \qquad \kappa = \nabla \cdot \bar{n} \tag{5}$$

The original work of level set method in two-phase numerical simulation is referred to [18]. A review of such works can be found in the cited review [19].

4. Breaking Dam Problem

In order to investigate the present problem, a two-dimensional broken dam problem is carried out. This test case, although of its simplicity in the initial configuration and the boundary conditions, it is useful for demonstrating the versatility of the developed numerical method. In this example, a rectangular hydrostatically equilibrium water column surrounded by air is allowed to flow out along dry horizontal floor, as shown in Fig. 1. The pressure jump condition at the interface is due to the gravity force acting downward, while the surface tension effect is neglected.

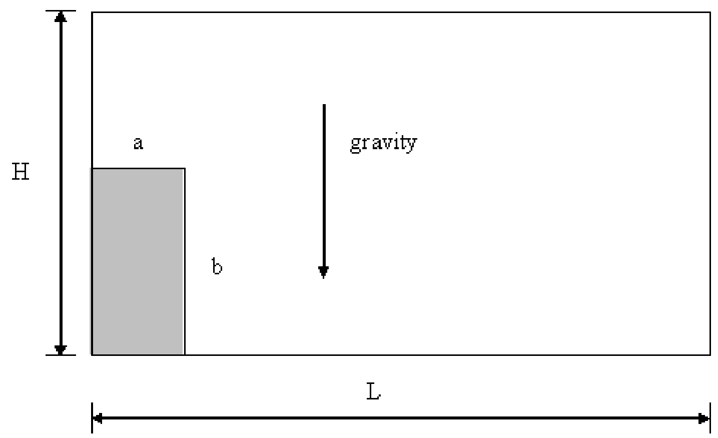


Figure 1. *Configuration of broken dam problem.*

The rectangular computational domain with size (L=1m, H=0.5m) is surrounded by solid walls, at which the free-slip conditions are imposed. The water column is 0.157m in width and 0.314m in height, and it has the density of 1000 kg/m^3 and the viscous coefficient of 1.8′10^{-3} Ns/m, while the air has the density of 1.0 kg/m^3 and the viscous coefficient of 1.5′10^{-5} Ns/m. The computational domain has uniform grid size in x- and y- directions, respectively. The water surface tension s is considered to be s= 0.07 N/m^2.

5. Numerical Procedure

In the current numerical method, a general differential momentum equation for the dependent variables (*u, v*) is written for unsteady, Newtonian, two-dimensional and incompressible flow:

The above level set equation as following equation:

$$\frac{\partial}{\partial t}(\rho\varphi)+\frac{\partial}{\partial x}(\rho u\varphi)+\frac{\partial}{\partial r}(\rho v\varphi)=\frac{\partial}{\partial x}(\Gamma_\varphi\frac{\partial\varphi}{\partial x})+\frac{\partial}{\partial r}(\Gamma_\varphi\frac{\partial\varphi}{\partial r})+S_\varphi \tag{6}$$

where φ is the dependent variable, Γ_φ is the diffusion coefficient for φ, and S_φ is the source term. The quantities Γ_φ and S_φ are specific to a particular meaning of φ. Using the control volume arrangement proposed by [20], the above general differential equation can be written in terms of the total fluxes over the control volume faces and the resulting equation is integrated over each control volume. In similar manner, the continuity equation is integrated over the control volume.

These equations relate the unknown value of dependent variable φ at time $t+\Delta t$ located at the pole of the control volume (P) to those neighboring points (E, W, N, S) and the known value of φ_P^o and ρ_P^o at time *t*. The source term S_φ is linearized by splitting it into S_C which stands for the constant part of S_φ, and S_b the coefficient of φ.

In our algorithm, one can presume that the velocity field reaches its final value in two stages; that means

The above level set equation as following equation:

$$\mathbf{u}^{n+1}=\mathbf{u}^*+\mathbf{u}_c \tag{7}$$

whereby, $\mathbf{u}^*$ is an imperfect velocity field based on a guessed pressure field, and $\mathbf{u}_c$ is the corresponding velocity correction. Firstly, the 'starred' velocity will result from the solution of the momentum equations. The second stage is the solution of Poisson equation for the pressure.

The above level set equation as following equation:

$$\nabla^2 p_c=\frac{\rho}{\Delta t}\nabla\cdot\mathbf{u}^* \tag{8}$$

where p_c will be called the pressure correction. Once this equation is solved, one gets the appropriate pressure correction, and consequently, the velocity correction is obtained according to:The above level set equation as following equation:

$$\mathbf{u}_c = -\frac{\Delta t}{\rho}\nabla p_c \tag{9}$$

The fractional step non-iterative method described above ensures the proper velocity-pressure coupling for incompressible flow field.

In order to solve the appropriate Poisson equation for pressure, one should assign the normal and tangential stresses resulting from the existing of the surface tension force. These stresses must be calculated exactly at the *interfacial marker points*; therefore, a suitable interpolation technique should be implemented. Given as example; the calculation of the local curvature at the marker positions, the average interface curvature as a function of the level set function ϕ and its derivatives can be expressed at the regular grid points according to the formula;

$$\kappa(\phi) = \frac{(2\phi_x\phi_y\phi_{xy} - \phi_x^2\phi_{yy} - \phi_y^2\phi_{xx})}{(\phi_x^2 + \phi_y^2)^{1.5}} \tag{10}$$

The present surface tension model ensures that both the pressure calculated within the liquid phase and the surface tension pressure is consistent and dynamically similar, as their effect is determined in the same way. Accordingly, the pressure drop across the interface cancels exactly the surface tension potential at the interface. Such modeling could, to some extent, reduce the spurious current problem that might be associated with the inconsistent modeling of the surface tension force in two-phase flow simulation, as the interfacial curvature and, consequently, the interfacial driving forces are calculated accurately.

In two-phase flow problems, the interface shape may be of complicated geometry. In such context, the accurate solution of the surface pressure occurring at transient fluid interfaces of arbitrary and time dependent topology is a key element of the proposed method, which involves a special treatment of the discretization at the interface.

Assuming that a regular mesh is used for the calculation, the curved shape of the interface causes unequal spacing between the interface and some internal grid points, the Dirichlet pressure boundary conditions are assigned at the interfacial marker positions for the calculating of the interfacial liquid pressure according to the case considered. The source term of Poisson equation is calculated at the main grid points using the imperfect velocity field calculated from the momentum equations and approximated at the irregular grid.

In order to solve Poisson equation, it should be first generalized for an arbitrary computational grid, in which the four neighboring points are at different distances from the central point of the control volume according to the interface position.

6. Results and Discussion

Figure 2 shows the present numerical results for the time histories of the horizontal water front location compared with both the experimental values presented in [2] and the numerical results obtained by using a novel finite-element technique [21]. The dimensionless time and displacement are given by $t^*=t(2g/a)^{0.5}$ and $x^*=x/a$. In the legend of this figure, the computational results are in very good agreement with the experimental ones.

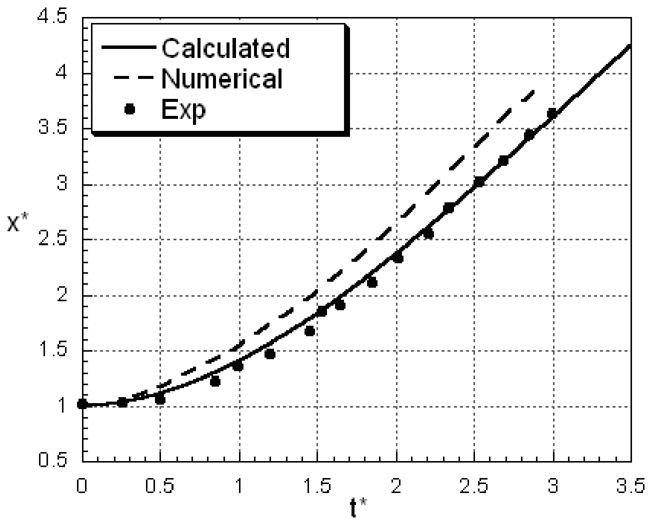


Figure 2. *Comparison of calculated results with the previous experimental measurements [2] and previous numerical results [21].*

Further, the effects of the ratio of water-to-gas density on the gravity force and consequently on the horizontal location of the water front are investigated and illustrated in Fig. 3. The computed results show that, minor difference could be observed when the density ratio between the two fluids is more than 1:100. However, the influence of different density ratios is clearly visible.

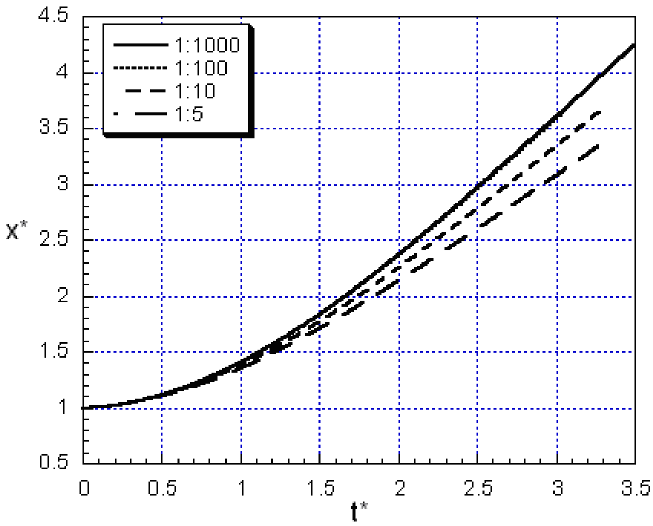


Figure 3. *Horizontal water front location for horizontal displacement of breaking dam at different density ratios.*

The figures below, figures 4 and 5, show the effect of water viscosity and surface tension on the horizontal movement of the breaking dam. It can be shown that by increasing the viscosity value and the surface tension, the front of the breaking dam is moved slower compared with increased values.

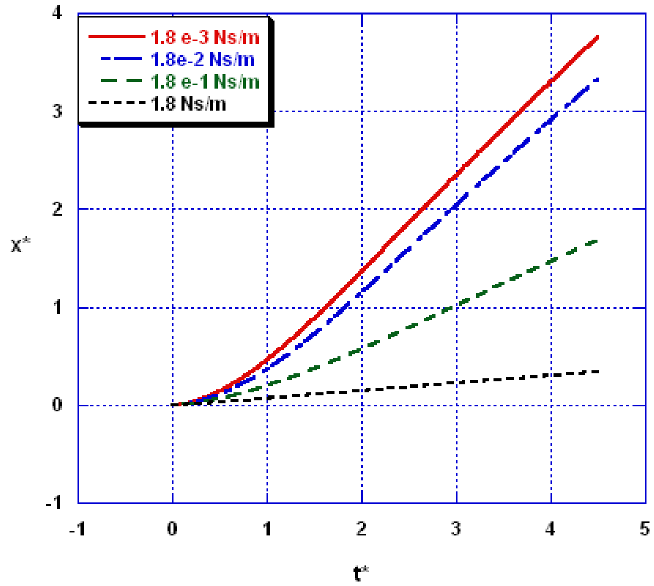


Figure 4. *Horizontal water front location for horizontal displacement of breaking dam at different water viscosity values.*

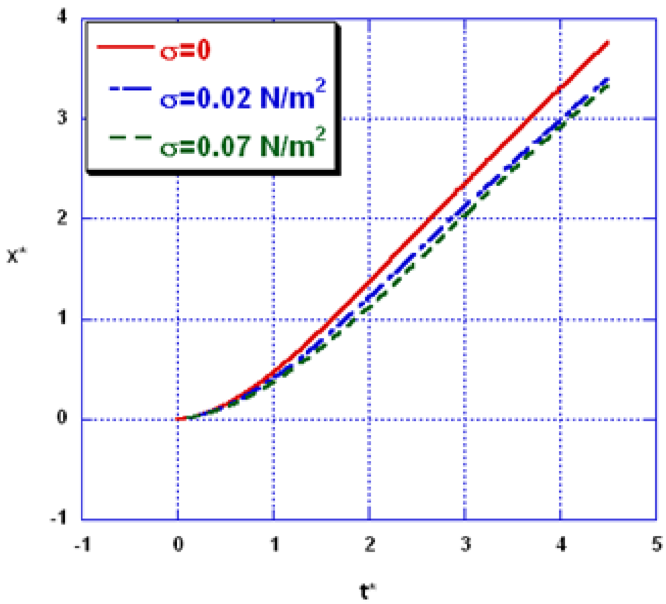
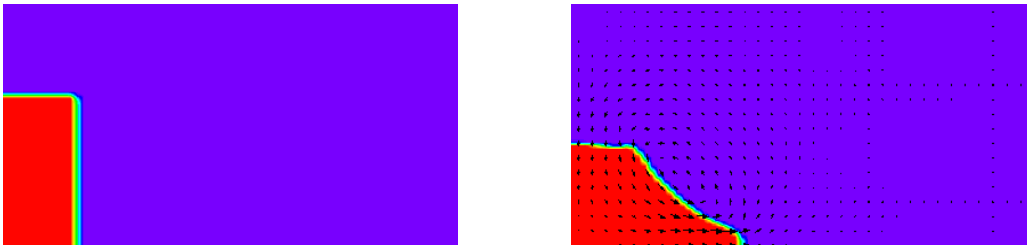


Figure 5. *Horizontal water front location for horizontal displacement of breaking dam at different water surface tension values.*

Figure 6 shows the computed fluid configurations and the flow field at several time levels for density ratio 1:1000. Obviously, the present method can successfully predict the topological changes of the interface and the formation of an entrapped air cavity.



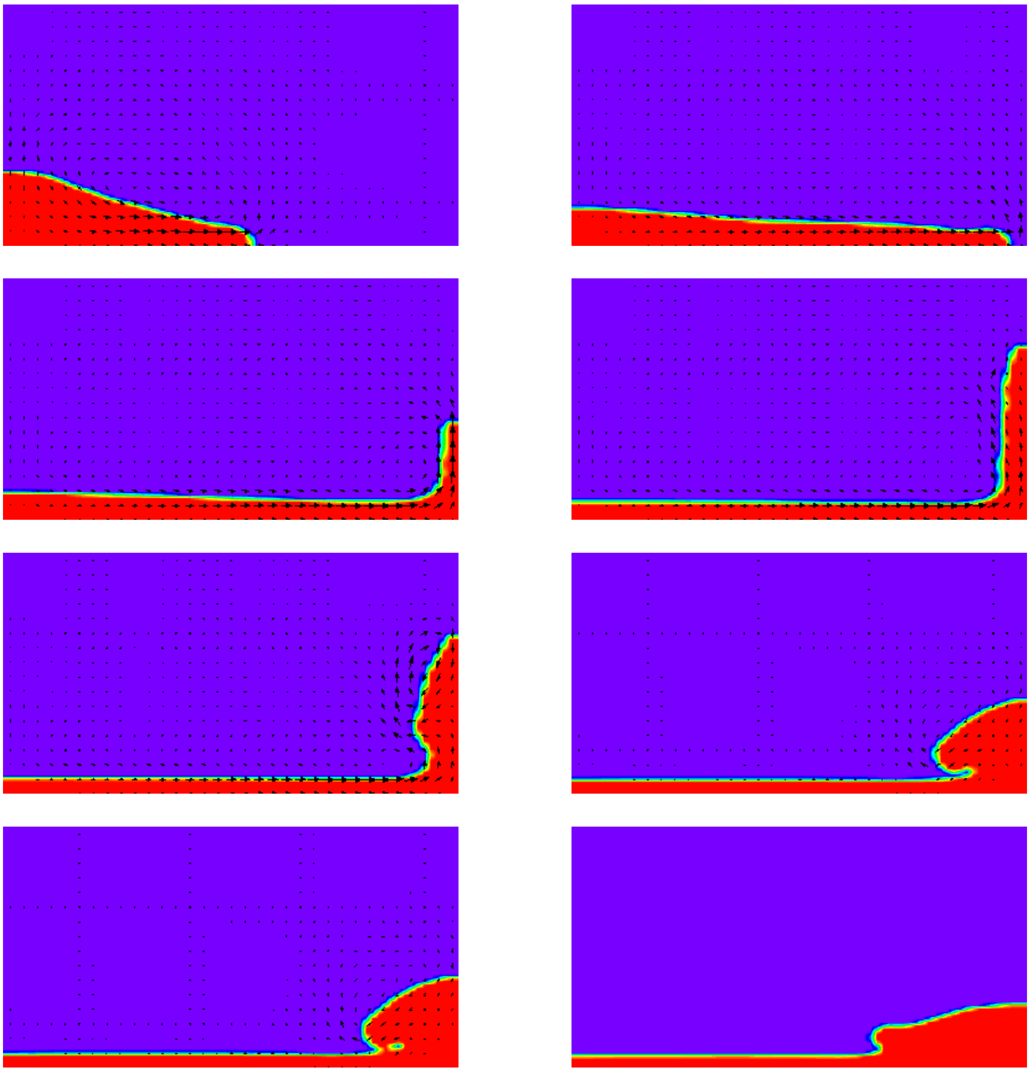


Figure 6. *Fluid configuration and flow field of broken dam at different times.*

7. Conclusions

In the present paper, the dam break problem is numerically simulated. The numerical method is based on solving the system of Navier-Stoke's equations with a developed numerical method known as IMLS method developed by the present author in previous investigations. The tracking and capturing of the dam free surface is carried out by the level set method. The comparison of the obtained numerical results showed a good agreement with the previous experimental measurements better than previous numerical results from the previous publications. The effect of fluid properties, such as density, viscosity, and surface tension is investigated. Generally, it can be concluded that the implementation of the developed numerical method in other two-phase flow applications can be straightforward and the extension of the numerical model to include a wide range of industrial and engineering applications could be easily task.

Conflicts of Interest

The authors declare that there is no conflict of interest regarding the publication of this article.

References

[1] Yang, C.; Lin, B.; Jiang, C.; Liu, Y. Predicting near-field dam-break flow and impact force using a 3D model, *J. Hydr. Research*, 2010, vol. 48 (6), pp. 784-792.

[2] Martin, J. C.; Moyce, W. J., An Experimental Study of the Collapse of a Liquid Column on a Rigid Horizontal Plane, Phil. Trans. Royal Soc. London 1952, A244, pp. 312-324.

[3] Koshizuka, S.; Oka, Y., Moving particle semi-implicit method for fragmenta- tion of incompressible fluid. *Nucl. Sci. Eng.*, 1984, vol. 123, pp. 421–434.

[4] Stansby, P.K.; Chegini, A.; Barnes, T.C.D. The initial stages of dam-break flow. *J. Fluid Mech*. 1998, 374, 407–424.

[5] Oezgoekmen, T. M.; Iliescu, T.; Fischer, P.F.; Srinivasan, A.; Duan, J. Large eddy simulation of stratified mixing and two-dimensional dam break problem in a rectagular enclosed domain. *Ocean Modelling*, 2007, vol. 16, pp.106–140.

[6] Park, R.; JinKim, K.; Van, S. Numerical investigation of the effects of turbulence intensity on dam-break flows, *Ocean Engineering*, vol. 42, pp. 176–187, 2012.

[7] Eggers, J. Nonlinear Dynamics and Breakup of Free-Surface Flow, Rews. *Modern Phys*, 1997, vol. 69, No. 3, pp. 865–929.

[8] Balabel A. Numerical Modelling of Liquid Jet Breakup by Different Liquid Jet/Air Flow Orientations Using the Level Set Method, *Computer modeling in engineering & sciences*, 2013, vol. 95 (4), pp. 283-302.

[9] Balabel, A.; El-Askary, W.; On the performance of linear and nonlinear k–e turbulence models in various jet flow applications. *European Journal of Mechanics B/Fluids*, 2011, vol. 30, pp. 325–340.

[10] Balabel, A., Numerical prediction of droplet dynamics in turbulent flow, using the level set method. International Journal of Computational Fluid Dynamics 2011, vol. 25, no. 5, pp. 239-253.

[11] Balabel, A. Numerical simulation of two-dimensional binary droplets collision outcomes using the level set method. *International Journal of Computational Fluid Dynamics*, 2012, vol. 26, no. 1, pp. 1-21.

[12] Balabel, A.; Numerical modeling of turbulence-induced interfacial instability in two-phase flow with moving interface. *Applied Mathematical Modelling*, 2012, vol. 36, pp. 3593–3611.

[13] Balabel, A. A Generalized Level Set-Navier Stokes Numerical Method for Predicting Thermo-Fluid Dynamics of Turbulent Free Surface. *Computer Modeling in Engineering and Sciences (CMES)*, 2012, vol.83, no.6, pp. 599-638.

[14] Balabel, A. Numerical Modelling of Turbulence Effects on Droplet Collision Dynamics using the Level Set Method. *Computer Modeling in Engineering and Sciences (CMES)*, 2012, vol. 89, no.4, pp. 283-301.

[15] Balabel, A; El-askary, W.; Wilson, S. Numerical and Experimental Investigations of Jet Impingement on a Periodically Oscillating-Heated Flat Plate. *Computer Modeling in Engineering and Sciences (CMES)*, 2013, vol. 95 (6), pp. 483-499.

[16] Shinjo J.; Umemura A. Simulation of liquid primary breakup: Dynamics of ligament and droplet formation", *Int. J. Multiphase Flow*, 2010, vol. 36(7), pp. 513-532.

[17] Hegab, A.; Gutub, S.; Balabel, A. A Developed Numerical Method for Turbulent Unsteady Fluid Flow in Two-Phase Systems with Moving Interface, International *Journal of Computational Method*, 2017, Vol. 14 (6), pp. 1750063-1-1750063-28.

[18] Sussman, M.; Smereka, P.; Osher, S. A level set approach for computing solutions to incompressible two-phase flows, *J. Comp. Physics*, 1994, 114, 146-159.

[19] Abu-alsaud, M.; Soulaine, C.; riaz, A.; Tchelepi, H. Level-set method for accurate modeling of two-phase immiscible flow with moving contact lines, *Fluid dynamics*, 2017, Vol. 2, pp.1-29.

[20] Patankar, S. V. Numerical Heat Transfer and Fluid Flow, Hemisphere Publishing Corporation, 1980.

[21] Kölke, A. Modellierung und Diskretisierung bewegter Diskontinuitäten in Randgekoppelten Mehrfeldaufgaben, Ph.D. Thesis 2005, TU, Braunschweig, Germany.