A New Numerical Method for Simulating Two-Fluid Interfacial Flow using Level Set Method

Ashraf Balabel
Mechanical Engineering Department, Faculty of Engineering, Taif University, Taif, Saudi Arabia.
(Email: ashrafbalabel@yahoo.com)

Abstract— The present paper’s technical feature describes a new numerical method for immiscible two-phase flow modeling and simulation on the basis of the level set method. The governing equations for time-dependent, two-dimensional and incompressible segregated flows are solved using the control volume approach on a non-staggered grid. The motion of each phase is treated separately, thereby; transition from one phase to another is performed through a consistent balance of kinematic and dynamic conditions on the interface separating the two phases. The topological changes of the interface are handled by the level set method, which simultaneously, provides an accurate technique for modeling the interfacial surface tension effects. The complete normal and tangential stress conditions on the interface are incorporated into the numerical technique, yielding a pronounced effect on low-Reynolds-number regimes. Some numerical test cases, that validate the accuracy and the capability of the proposed method- are presented. The method is then used to predict the ‘oscillatory deformations’ and ‘bag break-up’ regimes for a single droplet exposed in a convective gas flow. The obtained results demonstrate the performance of the numerical method adopted on many realistic problems of two-phase flow.

Keywords— Droplet dynamics, interfacial instability, level set method, numerical method, two-fluid flow.

I. INTRODUCTION

IMMISCIBLE two-phase problems are important and ubiquitous in nature, science and technology. Thereby a great attention has recently turned to the computational methods of two-phase flow dynamics and its realization in multi-space dimensions. On one hand, the reason for this increased attention lies in the tremendous technological importance of two-phase flow ranging from mixing to spraying, and chemical processing. On the other hand, the superior variability of carefully executed simulations with possible resolution in such fields can virtually replace experiments and provides us with an important source of information that is not available experimentally [1]. An extended review of recent advances in the computations of incompressible two-phase flows involving a fully nonlinear free surface that treated explicitly as a discontinuity can be found in [2].

Even though computational methods for two-phase flow dynamics have been developed rapidly in the last decades, there are still many hydraulic/thermal interfacial physical processes involved whose simulation is difficult to perform. These physical processes lead to a small-scale motion arising on/near the interface due to surface curvature and the associated interaction between the surface motions and the underlying flows. Viscosity and surface tension (and surfactant) effects are often important and responsible for linear/nonlinear surface motions during these processes. This includes many important examples such as hydrodynamic stability [3], drop oscillation [4], the dynamics of liquid bridge [5] and surfactant behaviour [6]. Detailed analysis of these interfacial processes with numerical models may aid in understanding the physical processes of multi-phase turbulent flows particularly with the recent progress in computer performance.

Generally, the numerical simulations of two-phase flow could become quite challenging problem and face significant difficulties. In particular, an accurate tracking of complex interfaces, where large deformations and inter-penetration of phases occur, is required. In addition, the basis of improving such numerical simulations lies in satisfying the interface- normal and tangential stress boundary conditions more accurately.

Crude approximations to the stress boundary conditions for free surface incompressible flow were used in the original Marker-and-Cell (MAC) computing method [7]. Later, the complete normal stress conditions were better approximated and incorporated in MAC method [8]. However, they were not always applied at the correct location of the interface. That can lead to a relative irregularity at the free surface demonstrating that a slightly spurious velocity is induced on the free surface. Following, an easy scheme was utilized in [9] to apply the correct normal and tangential stress conditions at the exact fluid surface location. However, the tangential stress is applied through the assignment of appropriate velocities near the interface by using a linear interpolation/extrapolation technique from the known velocity values at the main grid points. Furthermore, there are unique numerical instabilities are observed, which associated with the surface pressure interpolation technique.
Another approach for applying the appropriate boundary conditions at the free surface is developed in [10]; where the physical domain is transformed to a rectangle by means of a numerical mapping technique. This approach suffers from the complexity of determining the free surface position.

In order to describe the interface position and its complex topological changes with an elegant, robust and efficient method, the level set approach is applied [11, 12]. The level set formulation employs a smooth signed distance function, which is assumed to be positive inside one phase and negative outside. Consequently, an entire family of contours corresponding to the value of the level set at each grid point is obtained. The interface is described as the zero level set. According to the normal velocity field the level set function is updated at each grid point. The most attractive features of the level set approach is that it can automatically deal with breaking and merging of the interfaces and is naturally extendable to any number of space dimensions if the mass conservation is achieved.

The level set formulation presented in [11, 13] allows large density and viscosity ratios as well as surface tension to be included. However, this formulation considers that the effect of surface tension is to balance the jump of the normal stress along the fluid interface between inviscid fluids having a constant surface tension coefficient. This model takes the basic balance law of the continuum method [14] as a starting point, where the jump condition is reduced to Laplace's formula [15] for the surface pressure. Consequently, the tangential stress condition is not applied at the interface. The effect of neglecting the interfacial shear stress can be seen through increasing of the normal stress magnitude at the phase boundary [16, 17]. Besides, the capillary ripples, formed on a thin liquid film, receive their energy essentially from the pressure and shear stress exerted at the wave surface by the flowing concurrently high velocity gas stream [18].

In real flow, the satisfying of the interface stress conditions is a complex task involving significantly molecular interaction processes. According to [19], it is impossible with current computing capabilities to simulate both the large scale high Reynolds flow around a droplet and the molecular interactions. In this sense, an improvement of the stress boundary conditions for immiscible two-phase flow is required.

Most of the previous computational methods for immiscible two-phase flow are based on the treatment of the two phases as a continuous medium. Although this treatment does not need to employ any complex interface boundary conditions, a numerical instability under high-density ratio can be observed. Moreover, a numerical diffusion resulting from the smoothing of the tracked variables is obtained. To reduce considerably such numerical instability, a modified procedure based on the level set approach has been presented in [11]. This procedure considers an inviscid external fluid, usually taken to be a gas, in the surface stresses balance. The only influence of the gas is the pressure, it exerts on the liquid surface, which at the same time is considered as a reference pressure.

In general, all numerical methods for two-phase flow are developed under specific sets of assumptions, conditions and degrees of approximation. In this sense, a truly general numerical method is not available. Therefore, it is desirable to develop a computational method for immiscible two-phase flow, which can directly take account of the interfacial processes, and accurately capture the evolution of topologically complex moving interface with less computational effort.

The present article is organized in the following manner: section 2 describes the computational methodology for two-phase segregated flow. Starting from the basic governing equations, emphasis is placed on the formulation of the stress boundary conditions at the interface and the surface tension modeling. The evolution of the interface through the level set method is also described. Section 3 describes briefly the essential features of numerical procedure developed for solving the appropriate system of equations. In section 4 some numerical test cases, which provide the basis to evaluate the accuracy and the range of applicability of the proposed method, are presented. In section 5, one of the most challenge problem in two-phase flow is simulated; namely, the deformation, oscillation and breakup of single droplet. Finally, in section 6, the conclusion for the present work is drawn.

II. COMPUTATIONAL METHODOLOGY

A. Governing Equations

The governing transport equations for two-phase flow consisting of a gaseous flow around a liquid droplet can be represented by the continuity and momentum equations at each point of the flow field. Density is considered to be constant following fluid particle. According to the phase considered, the appropriate kinematic viscosity must be assigned to each point in the flow field. Presented here are the incompressible Navier-Stokes and continuity equations for two-dimensional flow in the constant density and viscosity form:

\[ \nabla \cdot \mathbf{u}_\alpha = 0, \]  
\[ \frac{D\mathbf{u}_\alpha}{Dt} + \nabla \mathbf{\tau}_\alpha = \nu_\alpha \nabla^2 \mathbf{u}_\alpha \]  

where \( \mathbf{u} \) is the fluid velocity vector, \( D/Dt \) is the material derivative, \( \mathbf{\tau} \) is the ratio of pressure to constant density, and \( \nu \) is the kinematic viscosity. The subscript \( \alpha \) is referred to either the gas or the liquid phase. The above
system of equations is solved initially either for the gaseous field or for the liquid phase, depending on the case considered, to obtain the velocity and pressure around the liquid phase or inside it. The solution of one phase is then used as interface boundary conditions for solving the other phase field. Next the deformation of the liquid is obtained for the given complete flow field dynamics. Since the velocity and pressure field inside the liquid phase are related to the external gas field tangential and normal stresses, a dynamic and kinematic balance between the gas flow field and liquid flow field should be made.

### B. Interfacial Boundary Conditions

The boundary conditions between the two immiscible fluids are denoted by the stress tensor equation at each point in the two-fluid flow as follows [20];

\[ \Pi_i = -p \delta_{ij} + \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \delta_{ij} \frac{\partial u_k}{\partial x_k} \]  

(3)

where \( \delta_{ij} \) is the Kronecker delta function, \( u_i \) represents the components of the velocity vector, \( x_i \) represents the components of the position vector; \( \mu \) is the dynamic viscosity, and \( p \) is the pressure. The above equation is generally applicable in fluid flows; however, this equation loses its generality on the interface between the two phases because of the discontinuous fluid properties. Let \( \Gamma \) denote the fluid interface. The condition of a balance of the forces at the interface may be written as a vector equation [14, 15];

\[ [p_1 - p_2 - \sigma \kappa] n_l = (\tau_{1,i} - \tau_{2,i}) n_j + \frac{\partial \sigma}{\partial x_i} \]  

(4)

where \( p_1, p_2, \tau_{1,i}, \tau_{2,i} \) are the pressures, viscous stress tensors, in phases 1 and 2, respectively, \( \sigma \) is the surface tension, \( \kappa \) is the curvature of the surface, \( n_l \) are the components of the unit normal vector from the surface directed into the interior of phase 2. In the case of an incompressible flow, the viscous stress tensor is given by:

\[ \tau_{ij} = \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \]  

(5)

In order to fix ideas, the subscripts \( l \) and \( g \) are used for the liquid and gas phase respectively. Taking the projections of the above vector equation, Eq. (4), in the directions normal and tangential to the interface, one can obtain, correspondingly, the two scalar conditions at the interface as given in [21];

\[ p_l - p_g - \sigma \kappa = 2 \mu \left( \frac{\partial v_l}{\partial n} \right)_l - 2 \mu_g \left( \frac{\partial v_g}{\partial n} \right)_g \]  

(6)

for the normal direction, and

\[ \mu_l \left( \frac{\partial v_l}{\partial t} + \frac{\partial v_l}{\partial n} \right)_l - \mu_g \left( \frac{\partial v_g}{\partial t} + \frac{\partial v_g}{\partial n} \right)_g = \frac{\partial \sigma}{\partial t} \]  

(7)

for the tangential direction, where \( \hat{t} \) designates the direction tangential to the interface, and \( v_l, v_g \) are the velocity components in the normal and tangential directions. The normal and the tangential stress boundary conditions are

\[ \frac{\partial}{\partial n} = n \cdot \nabla, \quad \frac{\partial}{\partial t} = \hat{t} \cdot \nabla \]  

(8)

One can split the velocity components \( u \) in the direction of the normal and the tangential of interface. Mathematically, this can be written as:

\[ \begin{pmatrix} v_n \\ v_t \end{pmatrix} = \begin{pmatrix} n_i n_j \\ -n_i n_j \end{pmatrix} \begin{pmatrix} u_l \\ u_g \end{pmatrix} \]  

(9)

Accordingly, the normal and tangential stress boundary conditions can be simplified in terms of Cartesian velocity components and the normal vectors. According to the above boundary conditions, it is evident that the normal stress boundary condition can be satisfied at the interface between two phases that are at rest, while the presence of tangential strain must be accompanied by convective motion, which must take place in at least one of the phases. It can be seen also that spatial variation in the surface tension coefficient along the interface can lead to a fluid motion in the system [2]. The motion induced by tangential gradient of surface tension is called the Marangoni effect. In the types of problems that we shall encounter, the stress boundary conditions is considered for two-dimensional surface with constant surface tension, the tangential stress is continuous across the interface since the jump in it is zero.

### C. Modeling of Surface Tension

The surface tension effect is modeled as boundary values of pressure present only at the interface. The equation describes the normal stress boundary conditions at the interface, Eq. (6), is used to calculate the pressure jump across the interface. In case of neglecting the viscous force, the pressure jump is calculated through Laplace’s formula [14, 15].

\[ p_l = p_o + \sigma \kappa \]  

(10)

where \( p_o, p_l \) are the pressure just inside and outside the interface separating the two phases. Obviously, the surface tension force at the interface between two immiscible fluids depends essentially on the prescribed curvature of the interface. By fitting a number of vertices
on the intersection points of computational grids with the interface, the local curvature is easily estimated. Once the curvature is known, the surface tension force is evaluated and then used to drive the liquid phase through the pressure gradient seen in the momentum equations.

This model ensures that, both the pressure calculated within the droplet and the surface tension pressure are 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. For more details about the proposed model and its numerical implementation, one can see [4].

### D. Level Set Formulation

In order to follow the evolution of an interface $\Gamma$ propagating with a defined speed normal to itself, the level set method is adopted. The essential idea of the level set is to construct a scalar distance function $G(\mathbf{x}, t)$ defined in all of the entire computational domain, such that the zero level set ($G=0$) always corresponds to the position of the interface $\Gamma$. That is,

$$\Gamma = \{ \mathbf{x} : G(\mathbf{x}, t) = 0 \}$$

(11)

By such formulation, the level set function is basically used to identify two separated regions with different properties. As a clear example are the incompressible two-phase flows. According whether $G>0$ or $G<0$, one can simply define both phases on both sides of the interface. Hence we have:

$$G(\mathbf{x}, t) = \begin{cases} > 0 & \text{ if } \mathbf{x} \in \text{ liquid} \\ = 0 & \text{ if } \mathbf{x} \in \Gamma \\ < 0 & \text{ if } \mathbf{x} \in \text{ gas} \end{cases}$$

(12)

and consequently,

$$u = \begin{cases} u_l & \text{ if } G > 0 \\ u_g & \text{ if } G < 0 \end{cases}$$

(13)

Obviously, through the application of these boundary conditions, the velocity components are continuous across the interface. The unit normal to the interface drawn outwards from the liquid to the gas, and also the curvature of the interface can easily be expressed in terms of the level set function $G$ as:

$$\mathbf{n} = -\frac{\nabla G}{|\nabla G|}, \quad \kappa = \nabla \cdot \mathbf{n}$$

(14)

For all time, the evolution of the level set can be described by the following equation which obtained from the chain rule:

$$\frac{\partial G}{\partial t} + \mathbf{u} \cdot \nabla G = 0$$

(15)

More precisely, one can rewrite the above level set equation as [23]:

$$\frac{\partial G}{\partial t} + F_{\text{ext}} |\nabla G| = 0$$

(16)

where $F_{\text{ext}}$ is some velocity field, known as the extension velocity, which at the interface equals the local normal speed. A variety of methodologies have been used to construct the extension velocity field in one form or another. Mainly, two approaches have widely been used in the two-phase flow simulations. In the first approach, one can choose to directly use the local fluid velocity itself to act as the extension velocity [11, 12]. However, the second one is based on the fast marching method developed in [22], where an extrapolation technique is used to build a speed function starting from the known velocity values at the interface. More details about the extension problem of the level set can be found in [23].

When the level set function is advected according to the above equations it will not remain a distance function for the points away from the interface (i.e. $|\nabla G| \neq 1$), and therefore it must be reinitialized to be a distance function [11]. In the present work, the reinitialization process described in [24] is used.

### III. NUMERICAL PROCEDURE

In the present study, a computational method such that the motion of each phase is separately treated is applied. The solution technique is based on an implicit fractional step type method. This method has an important advantage of not limiting the density ratio between the phases, while maintaining high numerical stability. The algorithm will only be sketched here for a uniform two-dimensional spatial grid.

#### A. Computational Grids

The governing equations for pressure, velocity components and level set function are discretized on a uniform Cartesian mesh using a non-staggered grid, see Fig. 1.

The computational grid points are classified into three sets. The main point is referred to the grid point at which all the relevant variables are defined and stored. Consequently, the phase change can be identified. All points, one grid point away from the interface are tagged as interface boundary points. Those points are considered as a moving boundary for the computational domain of both liquid and gas phase. Finally, the intersection points
are defined where the interface intersects the grid lines. Then, the surface curvature is evaluated at such points representing an accurate modeling of the surface tension effect between the two phases.

Our strategy for solving the appropriate system Eqs.(1 and 2) having two parts: first the advection, convection and the diffusion terms are solved for intermediate velocity field with a scheme nearly close to the one initially developed in [25] without strictly enforcing the incompressibility constraint. Second, the resulting velocity field is projected and the correct pressure field is calculated resulting in a divergence free velocity field. By solving the appropriate pressure correction equation, velocity correction is obtained.

B. Description of Algorithm

In this algorithm, a general differential equation, Eq. (17), for dependent variables (velocity components) is solved for unsteady, incompressible and two-dimensional \((\mathcal{I} = 0, r \equiv y)\) or axisymmetric \((\mathcal{I} = 1)\) under the appropriate boundary conditions:

\[
\frac{\partial}{\partial t} (\rho \varphi) + \frac{1}{r} \left( \frac{\partial}{\partial x} (\rho r^2 u \varphi) + \frac{\partial}{\partial r} (\rho r^2 v \varphi) \right) = \frac{1}{r^2} \left( \frac{\partial}{\partial x} (\Gamma_\varphi \frac{\partial \varphi}{\partial x}) + \frac{\partial}{\partial r} (\Gamma_\varphi \frac{\partial \varphi}{\partial r}) \right) + S_\varphi
\]

(17)

where \(\varphi\) is the dependent variable, \(\Gamma_\varphi\) is the diffusion coefficient for \(\varphi\), and \(S_\varphi\) is the source term. The quantities \(\Gamma_\varphi\) and \(S_\varphi\) are specific to a particular meaning of \(\varphi\). Using the control volume arrangement shown in Fig. 2, 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. By manipulating of the two integrated equations, one can derive the discretization equation:

\[
a_p \varphi + a_E \varphi_E + a_W \varphi_W + a_N \varphi_N + a_S \varphi_S + b = 0
\]

(18)

where:

\[
b = S_c \Delta x \Delta y + a_p^o \varphi_p^o
\]

(19)

and

\[
a_p = a_E + a_W + a_N + a_S + a_p^o - S_y \Delta x \Delta y
\]

(20)

and

\[
a_p^o = \frac{\Delta x \Delta y \rho_p^o}{\Delta t}
\]

(21)

These equations relate the unknown value of dependent variable \(\varphi\) 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 \(\varphi_p^o\) and \(\rho_p^o\) at time \(t\). The source term \(S_\varphi\) is linearized by splitting it into \(S_c\) which stands for the constant part of \(S_\varphi\), and \(S_y\) the coefficient of \(\varphi\). For more detailed descriptions, the reader is referred to [25].

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

\[
u^{n+1} = u^* + u_c
\]

(22)

whereby, \(u^*\) is an imperfect velocity field based on a guessed pressure field, and \(u_c\) 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

\[
\nabla^2 p_c = \frac{\rho}{\Delta t} \cdot u^*
\]

(23)

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:

\[
u_c = -\frac{\Delta t}{\rho} \nabla p_c
\]

(24)

The fractional step non-iterative method described above ensure proper velocity-pressure coupling for incompressible flow field.
C. Tangential Stress Modeling

Attention is now directed to the incorporation of the interfacial tangential stress. The interface can take several possible shapes that need to be considered when it intersects the control volume of \( u \)- or \( v \)-velocity, see Fig. 3. According to the location of the interface between two similar control volumes, the projected area of the interface is calculated and the tangential stress is considered to act upon it. The computational error, resulting from the replacement of the true length of the interface through its projected area can be reduced by using considerable fine grid resolutions.

For each control volume, the tangential velocity for the interface boundary points is obtained from usual momentum balance. The correct shear force expression is now inserted from Eq. (7) and incorporated via the source term treatment. The assumption of using the projected area of the interface simplifies Eq. (7), in such a manner that the gradient of velocity components in direction of the projected area is vanished and only the gradient perpendicular to it is considered. Mathematically, this implies that the shear strain rate exerted on the \( u \)- and \( v \)-control volumes can be expressed as:

\[
\varepsilon_x = (n_y^2 - n_x^2) \frac{\partial u}{\partial y} - 2n_y n_x \frac{\partial v}{\partial y} 
\]

\[
\varepsilon_y = (n_y^2 - n_x^2) \frac{\partial v}{\partial x} + 2n_y n_x \frac{\partial u}{\partial x} 
\]

The above equations should be calculated at the actual fluid surface. To accomplish this, the local properties of the interface (normal vectors and curvature) are specified at the intersection points as a linear interpolation between two adjacent interface boundary points, see Fig. 4.

Referring to Eq. (7), and assuming that the surface tension coefficient is constant, the jump of the shear stress across the interface is vanished. Therefore, the shear stress at the interface can be either calculated from the gas or liquid side. For refined computational grids, we can simply assume a linear variation of velocity profiles between any intersection point and the nearby interface boundary points. Consequently, from Fig. 4, the shear stress at the interface can be expressed as (Taking \( \mu = \frac{\partial v}{\partial x} \) for simplicity):

\[
\tau_i = \mu \frac{v_p - v_e}{\Delta x_i} = \frac{\mu}{\Delta x_g} \frac{v_e - v_W}{\Delta x_g} 
\]

where \( \tau_i \) is the interfacial shear stress located at the intersection point, \( \Delta x_i \) and \( \Delta x_g \) are the distances between the intersection point and the interface boundary points on the liquid and gas side respectively. The velocities \( v_p, v_e, v_W \) are located at the interface boundary points P, W and the intersection point e,
respectively. Eliminating the interface velocity $v_e$ from the above expression of the shear stress, one can obtain:

$$
\tau_i = \frac{\mu_i \mu_g}{\mu_i \Delta x_g + \mu_g \Delta x_i} (v_p - v_W) \quad (28)
$$

or in other form:

$$
\tau_i = \frac{\mu_i v_p - v_W}{\Delta x} \quad (29)
$$

where $\mu_i$ is the interfacial dynamic viscosity expressed as:

$$
\mu_i = \frac{\mu_i \cdot \mu_g}{\mu_i (1 - f_e) + \mu_g f_e} \quad (30)
$$

and $f_e$ is an interpolation factor defined in terms of distances:

$$
f_e = \frac{\Delta x_i}{\Delta x} \quad (31)
$$

When the intersection point is located midway between the interface boundary points, i.e., $f_e = 0.5$ then the interfacial dynamic viscosity is calculated as:

$$
\mu_i = \frac{2 \mu_i \mu_g}{\mu_i + \mu_g} \quad (32)
$$

which implies that $\mu_i$ is the harmonic mean of the liquid and gas viscosity. This is consistently with the derivation described in [25] for calculation of the interface diffusion coefficient used in the general differential equation, Eq. (17). In such a manner, the normal stress boundary conditions are calculated and incorporated in the solution of the Poisson equation at the actual fluid surface, i.e. at the intersection points.

IV. VALIDATION OF NUMERICAL METHOD

In order to check the fundamental performance of the developed numerical method, several illustrative examples of unsteady two-phase flow, ranging from simple to complex interface topologies, are simulated.

A. Fully-Developed Coutte Flow

Here is considered a fully-developed Coutte flow for two immiscible fluids between two flat plates or through an axisymmetric channel. The complete features of both problems are shown in Fig. 5. The appropriate governing equations for both cases are solved for the velocity distribution and the results are graphed and compared with the exact solutions using representative values of fluids properties. The purpose of this case is to illustrate in what extent the discontinuity in the velocity gradient between the two fluids is correctly captured. The exact solution of the 2D/axisymmetric problem considered, assuming fully developed flow $L >> b$, is given, respectively, by:

$$
\begin{align*}
\frac{2y}{b} \frac{\mu_2}{\mu_1 + \mu_2} U & \quad 0 < y \leq b/2 \\
\left(\frac{2y}{b} \frac{\mu_1}{\mu_1 + \mu_2} - \frac{\mu_1 - \mu_2}{\mu_1 + \mu_2}\right) U & \quad b/2 < y < b
\end{align*}
$$

$$
\begin{align*}
U \ln\left(\frac{R}{R_1}\right) - \frac{\mu_1}{\mu_2} \ln\left(\frac{R_{11}}{R_2}\right) & \quad R < r \leq R_1 \\
U \ln\left(\frac{R}{R_1}\right) - \frac{\mu_1}{\mu_2} \ln\left(\frac{R_{11}}{R_2}\right) & \quad R_1 < r \leq R
\end{align*}
$$

The density is considered to be the same in both fluids and the pressure term may be omitted. Figure 6 indicates the steady state velocity profiles for different ratios of $\mu_1/\mu_2$. It is evident from Fig. 6 that the computational results give remarkably excellent agreement with the exact solutions in all cases.

B. Lid-Driven Cavity Flow

In order to show the general robustness of the flow solver the lid-driven cavity flow is solved for $Re=100$, and $Re=400$ in single-phase flows. This case is considered as a benchmark problem which can be used to assess the effectiveness of numerical methods to prevent velocity pressure de-coupling and to test the level of accuracy introduced by the convection terms treatment. The results are compared with previous results of [26].

Figure 7 shows the normalized horizontal and vertical velocity components on the vertical and horizontal lines of the square cavity, respectively. The flow field vorticity contours for $Re=100$ and $Re=400$ are shown in Fig. 8. It can be noticed that there is no discernible difference between the predicted velocity profiles and the previous ones. The accurate prediction of the flow field velocity
profiles and consequently the associated vorticity contours demonstrates the capability of the present code to handle a wide range of incompressible fluid dynamics problems.

V. NUMERICAL RESULTS

In the present section, one of the most important problems in two-phase flows are investigated; namely, the droplet dynamics including deformation and disintegration.

A. Droplet Deformation

The presented numerical procedure is used to simulate the oscillatory deformation and disintegration of a two-dimensional circular droplet exposed in a convective gas flow. Both liquid and gas are solved simultaneously.

The density and viscosity of the two mediums are assumed equal. Thus the gravity force is neglected. At each time step, surface pressure is calculated at each droplet surface vertex by satisfying the interface boundary conditions, and then level set method is used to describe the topological changes of the droplet surface at each time step.
As it known, the competing tendencies between aerodynamic forces and surface tension forces and their effects on droplet deformation can be related to the Weber number $We$, and the Reynolds number $Re$. Different basic droplet dynamic behaviors have been experimentally and numerically observed and classified using the criteria based on $We$ and $Re$ [27, 28]. Two cases have been considered for the simulations according to the established criteria, Case(1), shown in Fig. 9 where the initial Weber number $We_i=85$ and $Re_i=65$, should result no breakup and the oscillatory deformations mode is obtained. Case(2), shown in Fig. 10 where $We_i=339$ and $Re_i=133$, should result in "parachute" type breakup (bag mechanism). Typical velocity vectors plots and droplet surface shapes, represented by the level set contour ($G=0$), for Case(1) and Case (2) are illustrated at different time steps.

Figure 9, as the droplet surface changes from the equilibrium position, the surface tension force changes, consequently, along the droplet surface. The resulting surface tension restoring force is not small compared to the aerodynamic force (because of the relatively smaller Weber number), thus the droplet starts to oscillate. During the transition stages of oscillation, a typical third mode surface shape is observed confirming that the droplet oscillating primarily at the third harmonic. At later times, the droplet begins deforming back, but, due to the viscous effects, the amplitude of oscillation decays and the droplet reaches its equilibrium circular form. In this case, no break-up is obtained.

B. Droplet Disintegration

Figure 10, instead of oscillating, the droplet flattens in the axial direction due to the relatively larger Weber number. A non-uniform pressure distribution is developed on the droplet surface resulting in a continual deformation of the droplet. Consequently, a re-circulation zone is well established in the wake region of the droplet. As a result, the droplet edges are stripped away, so as to form a crescent (liquid disk) and the droplet starts to break off into two ligaments. Afterwards, the two ligaments formed reach the equilibrium circular form due to the internal restoring surface tension force. It can be seen that, the deformation and the 'bag break-up' regime map derived from the simulation results, is in quite good agreement with the previous numerical and experimental data. Furthermore, the current algorithm can simply predict the physical phenomena after the droplet disintegration occurs without any numerical complexities.

In general, the present developed numerical method has shown its capability to extend to include a large numbers of two-phase flow applications with moving interface even with complex dynamics associated with turbulent flow. This can be seen in our recent published papers [29-32].

VI. CONCLUSION

An accurate and robust numerical method for predicting the effects of the interfacial forces arising due to buoyancy or capillary effects in two-phase flow has been developed. The present numerical model interprets the surface and the body forces as a boundary value conditions for the pressure on the interface. The surface pressure formula works well for the interface discontinuity. The mathematical problem is formulated in primitive variables and solving using the projection method for non-staggered grid system. The level set method has been used to present the topological changes of the interface separating the two fluids. The accuracy has been checked by applying the numerical method to the calculation of capillary and sloshing waves. The numerical diffusion and dissipation are almost vanished in the adopted scheme. The validation of the developed numerical method is confirmed through its application to some numerical test cases and the results are compared with the available numerical and experimental results. The comparison indicates that the present method give
good agreement with previous experimental results and accurately satisfies the conservation of mass. The implementation of the proposed numerical method in three dimensional flows appears to be straightforward and the extension of the numerical models adopted to include many new and physically interesting problems could be easily done in the future.

REFERENCES