Movement of Micro-Particles at Fluid-Fluid Interface

Sham Bansal[1], Man Singh[2], Mohammad Amir[3], Ashish Kumar Garg[4], Jarrar Ahmed[5], Ishu Goyal[6]

[1] Delhi University, Department of Mathematics, Bharati College, C4, Janakpuri, New Delhi – 110058

[2] Delhi University, Department of Mathematics, SPM College (for women), West Punjabi Bagh, New Delhi - 110026

[3] Delhi University, Department of Mathematics, I.P. College for women, civil lines, New Delhi - 110054

[4] Delhi University, Department of Mathematics, JDM College, Rajinder Nagar, New Delhi - 110060

[5] Delhi University, Department of Mathematics, Dyal Singh College, Lodhi Road, New Delhi - 110003

[6]Panjab University Chandigarh, Department of Physics, Chandigarh - 160014

Abstract: ***The present work is an investigation of fluid-fluid two-phase flow past a cylinder. Ansys Fluent 13.0 software has been used to know the flow patterns. Volume of Fluid (VOF) technique is used to track the liquid-liquid interface. Volume of fluid multiphase flow approach is capable of predicting the overall performance of liquid-liquid two- phase flow. In the present work two immiscible liquids of different viscosities are flowing side by side in different phases. The flow patterns are related with the capillary and Reynolds numbers. The capillary and Reynolds numbers are quite small and therefore, flow is laminar and the shape of the interface between the immiscible liquids is controlled by the viscous forces and the interfacial tension. Therefore, viscous forces and surface tension forces are the main dominant forces in the present model. Therefore, the gravity force can be neglected because it has very less effect as compared to viscous and surface tension forces. So the gravity force has been neglected in the present work whereas the viscous force plays a dominant role.***

Keywords: Two-phase, Liquid-liquid flow, Volume of fluid model, Interface, Flow visualization.

1. Introduction

The subject of two-phase flow has an immense importance in various engineering systems for their optimum design and safe operations. The practical importance of two-phase flow lies in various modern engineering technologies related to nuclear energy, chemical processes and heat transfer systems. The foundation of two-phase flow formulation is given as the local instant formulation of the two-phase flow based on the single-phase flow continuum formulation and existence of the interface dividing the phases. The interfacial momentum transfer models are discussed in a great detail, because for most of the two phase flow systems, computational fluid dynamics is dominated by the interfacial structures and interfacial momentum transfer [1]. In two-phase flow analysis, the volume fraction is very important due to the two different fluids. Liquid-liquid flow system is one of the important experiments of two-phase flow system. The spatial structure of fluid-fluid flows can assume a variety of different configurations due to the deformable boundaries between the immiscible liquids. In a micro channel flow of Newtonian fluid, the Reynolds number is usually small due to the micro-scale dimensions of the flow passage, and flows of Newtonian fluid are limited to be laminar due to two factors: domination of viscous effects at small Reynolds number and the dominant interaction between the fluid and the channel wall at microscopic scale of the channel [2].

1.1 Governing Equations

In the present work, the equations used are the Incompressible Navier-Stokes Equations which comprise of the continuity equation and momentum equation.

Continuity Equation: The equation of continuity is a statement of mass conservation. Its general form is [3]

$$\frac{\partial \rho}{\partial t} + \nabla . \left(\rho \vec{v} \right) = 0 \tag{1}$$

Where ρ is density and $\vec{v}$ is velocity vector. Under the assumption of incompressibility, the density of a fluid is constant and it follows that the continuity equation [3] will simplify to:

$$\nabla . \vec{v} = 0 \tag{2}$$

Momentum Equation: A single momentum equation has been solved throughout the domain using volume of fluid model, and the resulting velocity field is shared among the phases. Because the fluids do not mix, each computational cell is filled with purely one fluid and purely another fluid or the interface between two (or more) fluids. Because of this unique set of conditions, only a single set of Navier-Stokes equations is required.

The momentum equation [4, 5] shown below, is dependent on the volume fractions of all phases through the properties μ and ρ.

$$\rho\left[\frac{\partial v}{\partial t}+\vec{v}\nabla.\vec{v}\right]=-\nabla p+\mu\nabla^{2}\vec{v}+\rho g+\vec{F} \qquad (3)$$

Where p is the pressure, μ is the viscosity; g is the gravitational force and F an additional body force.

In the present work, effect of gravity force has been neglected because it has very less effect compared to viscous and surface tension forces and Continuity and Momentum equations have been reduced to dimensionless form. New dimensionless variables have been defined:

$$\tilde{v}=\frac{v}{V}\ ,\quad \tilde{t}=\frac{\mu}{\rho L^{2}}t\ ,\quad \tilde{p}=\frac{L}{\mu V}p\ ,$$

$$\tilde{x}=\frac{x}{L}\ ,\quad \tilde{y}=\frac{y}{L}\ ,\quad \tilde{F}=\frac{L^{2}}{\mu V}F \qquad (4)$$

Therefore Continuity and Momentum equations in dimensionless form, become [4-6]

$$\tilde{\nabla}.\tilde{v}=0 \quad \text{and} \quad \frac{\partial\tilde{v}}{\partial\tilde{t}}=-\tilde{\nabla}\tilde{p}+\tilde{\nabla}^{2}\tilde{v}+\tilde{F} \qquad (5)$$

It is known as in stationary stokes equation.

2. Numerical Methodology

Numerical methodology in Fluent to solve the problem is quite good. It follows some algorithms to solve the problem. There are number of options of different methods that can be selected according to our model. Our problem is based on two phases. These two phases are called Primary and Secondary phases. Volume of fluid model uses the single set of momentum equations to solve the problem. The solution convergence depends upon the parameters and meshing. If either of parameters or meshing is not correct then solution may diverge or may give a poor convergence. The model equations have been solved using the commercial CFD software package Ansys Fluent 13.0. Fig.1 shows the general procedure for the simulation using Ansys Fluent software.

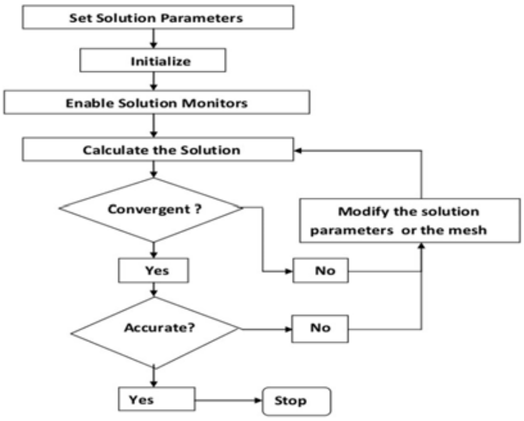

Figure 1: Flowchart showing the general procedure for the simulation using Fluent

2.1 Geometry and Mesh

The first step is to create the geometry, which has been done by DM (Design Modular) tools in Ansys, to design the problem in geometrical configuration and mesh the geometry. Before using Fluent, one has to solve the fluid flow problems; it needs the domain in which the flow takes place to evaluate the solution. The flow domains as well as the grid generation into the specific domain have been created in Design Modular. Two dimensional geometry has been created and meshed to form the grid. In order to create the desired geometry, firstly the surface body for the cylinder particle has been created, and then the surface body for the outer boundary was built. In two-dimensional geometry triangular and quadrilateral meshing techniques have been used to mesh the geometry. The boundary conditions such as velocity-inlet, pressure outlet, symmetry and default interior have been set. Then the grid has been exported as a mesh file from DM to be used in Ansys Fluent 13.0 for solution. Next, after meshing the geometry, mesh size is defined as: 11,826 elements, 18,442 faces and 12,504 nodes (See Fig. 2).

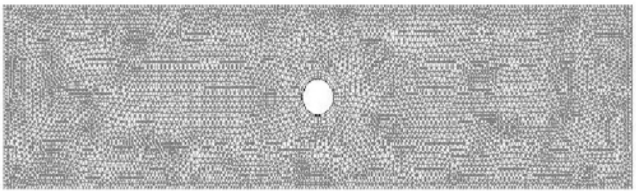

Figure 2: Mesh size: 11,826 elements, 18,442 faces and 12,504 nodes

2.2 Boundary and Initial Conditions

In present work, we have implemented reasonable boundary conditions for the computational domain. The diameter of cylinder particle has been taken as D = 200μm. The length of calculation domain is taken to be 20 times of the diameter, *i.e.* L = 4000μm *and* the height is considered as 5 times of the diameter, *i.e.* H = 1000μm. Inlet-velocity boundary condition for both the liquids is u_x = 0:001(*m/s*) and u_y = 0(*m/s*). Outlet boundary condition is pressure-outlet boundary condition, which has been set as 0 Pascal (*Pa*). Wall boundary condition is no-slip boundary condition, which has already defined. Boundary conditions are shown in Fig. 3

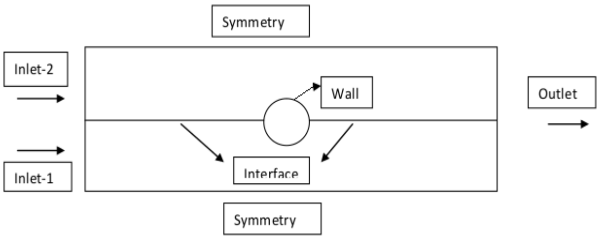

Figure 3: Calculation domain with boundary conditions

2.3 Solution Techniques

In Fluent, solver has been set as segregated which solves the equations individually. Unsteady state has been formulated as

1st order implicit condition and node based gradient option has been taken. The discretization scheme for momentum has been taken as second order upwind and the discretization scheme for volume fraction geometric reconstruction has been taken. The solution procedure involves the following steps:

- Generation of suitable grid system.
- Conversion of governing equation into algebraic equations.
- Selection of discretization schemes.
- Formulation of the discretized equation at every grid location.
- Formulation of pressure equation.
- Development of a suitable iteration scheme for obtaining a final solution.

For pressure-velocity coupling, phase Coupled Simple method [11] has been chosen. The solution has been initialized from all zones. For any set of convergence criteria, the assumption is that the solution is no longer changing with further iterations. In present work, the convergence criteria of all the residuals have been taken as 0.0001 and iterations have been carried out with time step size of 0.0005(*s*).

3. Results and Discussion

In the present work, velocity $U = 0.001(m/s)$, diameter of cylinder particle $D = 0.0002m$ and the viscosities of liquid-1 and liquid-2 have been taken in the range $0.001(kg/ms)$ to $0.0024(kg/ms)$. The densities of liquid-1 and liquid-2 have been taken in the range $780(kg/m^3)$ to $998.2(kg/m^3)$.

For this particular model, the dimensionless numbers are:

$$\mathrm{Re} \ll 1, \quad Ca \ll 1 \quad \text{and} \quad We \ll 1$$

Then inertial forces can be neglected, viscous forces play a dominant role and flow is laminar for such a Reynolds number. Surface tension forces also play important roles in the system as *Ca* and *We* are very low. Therefore viscous forces and surface tension forces are the main dominant forces in the present model. Therefore gravity force can be neglected because it is very less effective as compared to viscous and surface tension forces. So the gravity force in the present work has been neglected. The surface tension between two liquids has been taken as $\sigma = 0.001(N/m)$ and the contact angle, at which liquid-1 and liquid-2 interface meets the cylinder particle, has been taken as 90^0. Liquid-1 has been considered as water and liquid-2 can be any other liquid with different viscosity. In the present study, the ratio of viscosities of two liquids have been considered, which is defined as $\beta=\mu_2/\mu_1$ where μ_1 and μ_2 are viscosities of liquid-1 and liquid-2 respectively. Then three cases are possible.

1) $\beta = 1$ 2) $\beta > 1$ 3) $\beta < 1$

Case-I: When $\beta = 1$, all the parameters of both the liquids are same in this case. Liquid-1 is considered as water and therefore: $\rho_1 = \rho_2 = 998.2\ kg/m^3$ and $\mu_1 = \mu_2 = 0.001 kg/ms$ Where ρ_1 and ρ_2 are densities of liquid-1 and liquid-2 respectively. After running some time steps with time step size 0.0005 (*s*), it has been observed that there is no change in the shape of interface and fluid is moving smoothly over the cylinder particle as in single- phase problem. So when there is no change in flow then the calculation has been stopped after 2,000 time steps with time step size 0.0005 (*s*). See Fig.4, there is no change in the shape of interface.

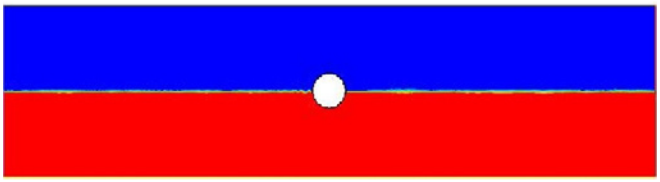

Figure 4: Contour volume fraction of liquid-1, after 2,000 time steps with step size 0.0005(*s*)

Case-II: When $\beta > 1$, for liquid-2, the viscosity has been taken as $\mu_2 = 0.0024\ (kg/ms)$. Therefore:

$$\rho_1 = 998.2(kg/m^3),\ \mu_1 = 0.001(kg/ms),$$
$$\rho_2 = 998.2(kg/m^3),\ \mu_2 = 0.0024(kg/ms)$$

In this case, after some iteration it has been observed that interface started moving towards liquid-1 because of the high viscous pressure of liquid-2 at the wall of cylinder particle near interface in the presence of interfacial force. Therefore interface started moving towards liquid-1. Viscous pressure difference of two liquids and capillary pressure are:

$$\Delta P_v = \frac{\mu_2 U_2}{R} - \frac{\mu_1 U_1}{R} \quad \text{and} \quad \Delta P_c = \frac{\sigma}{R} \qquad (6)$$

Where U_1 and U_2 are velocities of liquid-1 and liquid-2 respectively and R is radius of cylinder particle. Here, velocity of both liquids is same and $\mu_2 > \mu_1$. It implies there is some positive viscous pressure on the wall of cylinder near the interface and hence it started moving towards liquid-1, which is less viscous. Then the interface detaches from the cylinder particle (See figures 5, 6, 7 and 8)

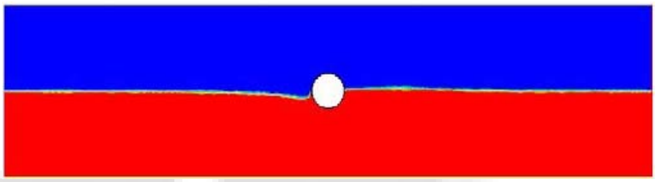

Figure 5: Contour volume fraction of liquid-1, after 5,00 time steps with step size 0.0005(*s*)

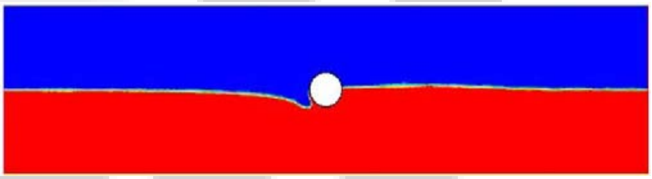

Figure 6: Contour volume fraction of liquid-1, after 1,000 time steps with step size 0.0005(*s*)

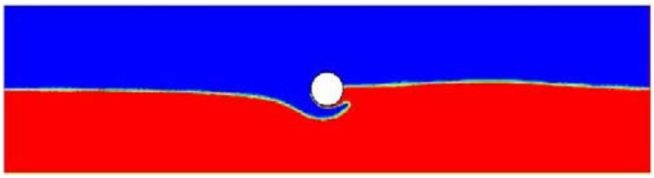

Figure 7: Contour volume fraction of liquid-1, after 2,000 time steps with step size 0.0005(*s*)

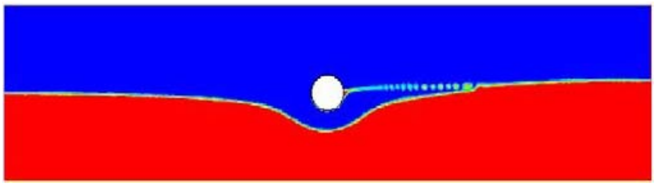

Figure 8: Contour volume fraction of liquid-1, after 5,000 time steps with step size 0.0005(*s*)

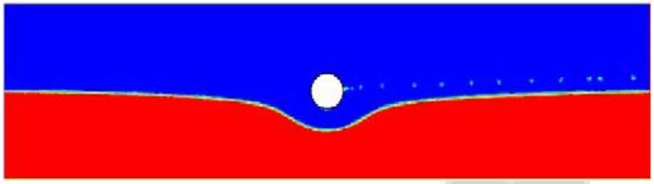

Figure 9: Contour volume fraction of liquid-1, after 10,000 time steps with step size 0.0005(*s*)

After running the 10,000 time steps with time step size 0.0005*s*, it has been observed that there is no more change in flow and in the shape of interface. It indicates that flow has come to quasi steady state. (Fig. 9)

Case-III: When $\beta < 1$, for liquid-1, the viscosity has been taken as

$\mu_2 = 0.0024\ (kg/ms)$. Therefore:

$$\rho_1 = 998.2(kg/m^3),\quad \mu_1 = 0.0024(kg/ms),$$
$$\rho_2 = 998.2(kg/m^3),\quad \mu_2 = 0.001(kg/ms)$$

After some iteration, it has been observed that interface started moving towards liquid-2 because of the high viscosity pressure of liquid-1 at the wall of cylinder particle near interface in the presence of interfacial force. It implies there is some negative viscous pressure on the wall of cylinder particle near the interface, so its direction is opposite now and hence it started moving towards liquid-2 and then the interface detaches from the cylinder particle. After running the 10,000 time steps with time step size 0.0005*s*, it has been observed that there is no more change in flow and interface. This implies that flow has come to quasi steady state. (See Fig.10)

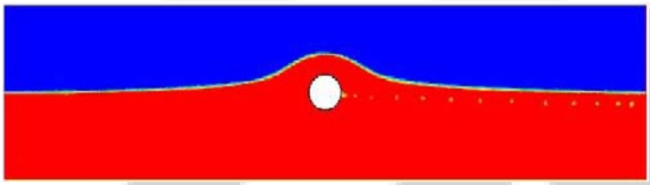

Figure 10: Contour volume fraction of liquid-1, after 10,000 time steps with step size 0.0005(*s*)

In another case, when densities of liquid-1 and liquid-2 are different and viscosities are same, then no effect of density has been observed and fluid flows smoothly over the cylinder particle as in single-phase problem.

Now, if $\beta > 1$ as mentioned in the Case-II, has been further investigated. Here, viscosity of one liquid has been fixed and the viscosity of other liquid has varied. Liquid-1 has been taken as water and liquid-2 can be any other liquid with different viscosity. Now consider: $\mu_1 = 0.001\ (kg/ms)$ and $\mu_2 = 0.0011\ (kg/ms)$. In this case, after some iteration it has been observed that there is small viscous pressure of liquid-2 on the wall of cylinder near interface in the presence of interfacial force and interface started moving towards liquid-1. In this case viscous pressure difference is very low. Therefore the interface covered the less space in phase-2. See Fig.11

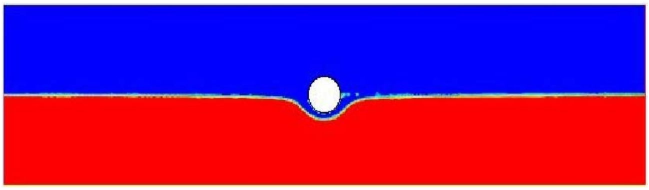

Figure 11: Contour of volume fraction for liquid-1, after 10,000 time steps with step size 0.0005(*s*), when $\mu_1 = 0.001$ *(kg/ms)* and $\mu_2 = 0.0011$ *(kg/ms).*

Now, the viscosity of liquid-2 has been changed to $\mu_2 = 0.0015\ (kg/ms)$. This time, it has been observed that there is little more viscous pressure of liquid-2 as compared to the case when $\mu_2 = 0.0011\ (kg/ms)$. It is because the viscous pressure is increasing, which creates more pressure on the cylinder wall near the interphase. Now liquid-2 covered little more space in phase-1.See Fig.12

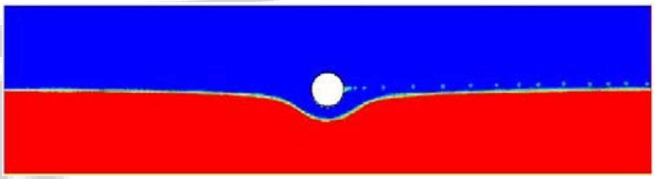

Figure 12: Contour of volume fraction for liquid-1, after 10,000 time steps with step size 0.0005(*s*), when $\mu_1 = 0.001$ *(kg/ms)* and $\mu_2 = 0.0015$ *(kg/ms).*

Hence, it is concluded that when the viscosity of liquid-2 has been increased then viscous pressure has increased on the wall of cylinder particle near the interface and liquid-2 covered more space in phase-1. The length between interface and the wall of cylinder particle (from bottom) has been taken as δ and this δ increases by increasing the viscosity of liquid-2. On this basis a table has been constructed, which represents the increment in δ with respect to viscosity. See Table1;

Table 1: Variation of length of δ with respect to increase in difference of liquid's viscosities and increment in length of δ per unit radius length of cylinder particle.

μ_1	μ_2	$\beta = \mu_2/\mu_1$	δ	δ/R
0.001	0.001	1	0	0
0.001	0.0011	1.1	3.35e-5	0.335
0.001	0.0012	1.2	4.65e-5	0.465
0.001	0.0013	1.3	5.75e-5	0.575
0.001	0.0014	1.4	6.65e-5	0.665
0.001	0.0015	1.5	7.40e-5	0.740
0.001	0.0016	1.6	8.10e-5	0.810
0.001	0.0017	1.7	8.75e-5	0.875
0.001	0.0018	1.8	9.40e-5	0.940
0.001	0.0019	1.9	1.00e-4	1
0.001	0.0020	2.0	1.055e-4	1.055
0.001	0.0021	2.1	1.105e-4	1.105
0.001	0.0022	2.2	1.150e-4	1.150
0.001	0.0023	2.3	1.195e-4	1.195
0.001	0.0024	2.4	1.235e-4	1.235
0.001	0.0024	2.5	1.265e-4	1.265
0.001	0.0026	2.6	1.295e-4	1.295
0.001	0.0027	2.7	1.325e-4	1.325
0.001	0.0028	2.8	1.350e-4	1.350
0.001	0.0029	2.9	1.375e-4	1.375
0.001	0.0030	3.0	1.400e-4	1.4

Table1 indicates that if viscosity difference is increasing then the increment length δ is also increasing, but the rate of increment of the length δ is decreasing. The increment in length δ per unit radius length of cylinder particle has been computed, which provides more clear view. The graph of $\beta = \mu_2/\mu_1$ and the increment length δ per unit radius length of the cylinder particle has been plotted in Fig.13

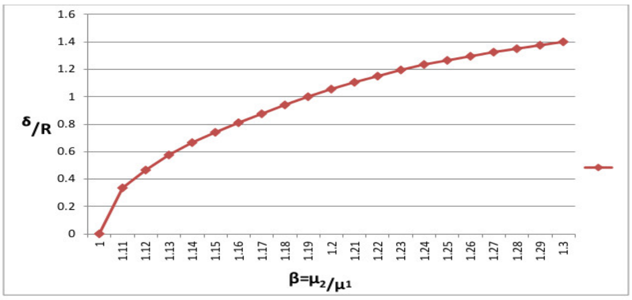

Figure 13: Graph of $\beta = \mu_2/\mu_1$ and the increment length δ per unit radius length of the cylinder particle.

From Fig.13 it is clearer that the increment length per unit radius increases with increase in difference of the viscosities. But it has been also observed from the graph that initially there is more increment in the length δ per unit radius of particle and the rate of increment decreases with the length of δ per unit radius of the particle.

In the present work, it has also been observed that all the results are grid independent. Firstly, the geometry has been meshed with 11,826 elements, 18,442 faces and 12,504 nodes and then the model has been simulated using this grid size. Then the mesh has been refined with mesh size 90,624 elements, 1, 37,020 faces and 93,000 nodes. Then the model has been simulated again using this grid-size. It has been observed that results are same with sharper interface. Results are not affected by increasing the grid size. It implies that results are grid-independent. On the basis of these two different grid sizes, contour of volume fractions of each of the cases with $\mu_1 = 0.001(\text{kg}/ms)$, $\mu_2 = 0.002(\text{kg}/ms)$ and $\rho_1 = \rho_2 = 998.2(\text{kg}/m^3)$ is shown in figures 14 and 15 respectively.

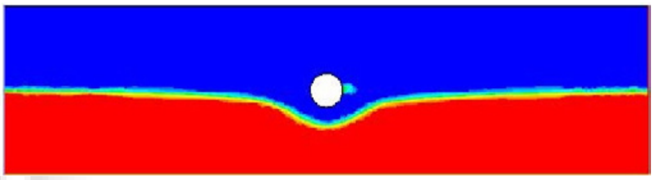

Figure 14: Contour of volume fraction for liquid-1 with mesh size 11,826 elements, 18,442 faces and 12,504 nodes, after running the 10,000 time steps with time step size 0.001(s)

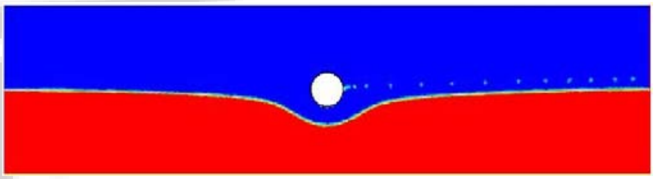

Figure 15: Contour of volume fraction for liquid-1 with mesh size 90,624 elements, 1,37,020 faces and 93,000 nodes, after running the 10,000 time steps with time step size 0.0005(s)

From figures, it can be observed that by increasing the number of elements interphase becomes sharper. But there is no difference in results; even the increment δ is same for all the three different grid sizes. Therefore results are grid-independent.

4. Conclusion

In this work, a report on movement of micro-particles at fluid-fluid interface has been studied. Results have been obtained by using volume of fluid (VOF) method in Ansys fluent 13.0 software. Results have been discussed on basis of different fluid properties. The dimensionless numbers *Re*, *Ca*, and *we* are very small compared to unity. Therefore inertial forces can be neglected. Viscous forces and surface tension forces are the main dominant forces in the model. Obtained are the following conclusions:

- Particle is always covered by high viscosity liquid.
- More viscous fluid moves towards the less viscous fluid in the presence of viscous pressure and on the wall of micro particle near the interface.
- When the viscosities of both the fluids are equal and densities are different, then there is no change in shape of interface and fluid flows in the same pattern as in the single-phase flow.
- When the fluids in two-phases have been interchanged, it has been found that the direction of interface also changes.

- When difference of viscosities of two fluids is less, then the more viscous fluid covers the less space in the less viscous fluid phase.
- When the difference of viscosities of the two fluids increased then the space covered by more viscous fluid in the less viscous fluid phase also increased.
- The rate of increment in space, covered by the more viscous fluid in the less viscous fluid phase, decreased by increasing the difference in viscosities of two fluids.
- Results are grid-independent, because after refining the mesh up to 90,624 elements, there is no change in the results.

Present study finds the applications in paints and surface coatings. The paints with higher viscosity than water can be used for Marine applications *i.e.* these results can be used to paint the ships to prevent corrosion in sea water. As in results, it has been mentioned that particle is always covered by high viscous liquid. Therefore, if viscosity of paint is higher than the viscosity of water then it will not allow the water to react on the surface.

5. Future scope of the work

In future, movement of particles can be tracked and forces acting on the particles can also be calculated. It may be applied for various other investigations and troubleshooting in industries.

References

[1] Mamoru Ishii, Takashi Hibiki, Thermo-Fluid Dynamics of Two Phase Flow, 2nd addition (Springer), 2011.
[2] Feng-Chen Li, Haruyuki Kinoshita *et al*, “Creation of very-low-Reynolds-number chaotic fluid motions in microchannels using viscoelastic surfactant solution”, Experimental Thermal and Fluid Science, 34, pp. 20-27, 2010.
[3] Pedlosky, Joseph, Geophysical fluid dynamics, Springer, 1987.
[4] Fox, Robert W., Alan T. McDonald, Philip J. Pritchard, Introduction to fluid mechanics (6th ed.), Hoboken, NJ: Wiley, 2006.
[5] Tritton, D.J., Physical fluid dynamics (2nd ed.), Oxford [England]: Clarendon Press, 1988.
[6] White, Frank M., Fluid mechanics (5th ed.), Boston: McGraw-Hill, 2003.
[7] Ansys Fluent 13.0 theory guide, Ansys, Inc., 2010.
[8] Bakker, A., Fluent introductory notes. Fluent Inc, Lebanon, 2002.
[9] J.U. Brackbill, D.B. Kothe, and C. Zemach, “A Continuum Method for Modeling Surface Tension”, J. Comput. Phys., 100, pp. 335-354, 1992.
[10] N. Brauner, D. Moalem Maron, “Stability analysis of Stratified liquid-liquid flow”, Dept. of Fluid Mechanics and Heat Transfer, Uni. of Tel-Aviv, Ramat-Aviv, Israel, pp.103-121, 1992.
[11] S. V. Patankar. Numerical Heat Transfer and Fluid Flow, Hemisphere, New York 1980.

Author Profile

Sham Bansal received the M.Tech. degree in Industrial Mathematics and Scientific Computing from Indian Institute of Technology Madras in 2012. He did M.Tech. Project in TU Darmstadt, Germany under DAAD Scholarship.