EFFECT OF INTERPHASE FORCES ON TWO-PHASE LIQUID: LIQUID FLOW IN HORIZONTAL PIPE

(Date received: 19.7.2007)

Rashimi, W., Choong, T. S. Y*., Chuah, T. G., Hussain, S. A. and Khalid, M.
Department of Chemical and Environmental Engineering Faculty of Engineering,
Universiti Putra Malaysia 43400 UPM Serdang Selangor
*E-mail: *tsyc2@yahoo.com.uk*

ABSTRACT

A two-fluid model (Eulerian-Elurian model) is used to simulate dispersed two-phase immiscible liquids (oil-water) in a horizontal pipe. Effect of interphase forces (drag, lift and turbulent dispersion) is discussed. In the present study water is considered as dispersed and oil as continuous phase. The exchange between the phases is represented using source terms in conservation equations. Standard k-e turbulence model is used to induce turbulence in continuous phase. Comparison between mathematical simulation using CFD code FLUENT 6.2 and experimental data indicates that the interphase forces are important and has a strong effect on flow behaviour. Different drag, lift and turbulent dispersion expressions are evaluated. The CFD simulations are in good agreement with published experimental data.

Keywords: *CFD, Dispersed Flow, Horizontal Pipe, Interphase Forces, Oil-water System, Two-phase Flow*

1.0 INTRODUCTION

Liquid-liquid dispersions comprising of drops of one fluid dispersed into other are extensively encountered in many of the chemical and process industries. An oil-water mixture flow patterns are unique and also complex due to its complicated rheological behaviour.

A quantitative and qualitative understanding of the flow hydrodynamics, turbulence and dispersion phenomena is necessary for optimal design and efficient operations of equipment in handling such liquid dispersions. Heat and mass transfer rates are also affected by dispersed entities

that feature varying shapes and size. Mass transfer rates are greatly enhanced if the interfacial contact area of the dispersed phase is increased.

Extensive numerical modelling of gas-liquid flow have been carried out and reported such as turbulence phenomena, fluid flow, mixing phenomena for flow in vertical pipe. However, the knowledge of gas-liquid flows cannot be readily applied for liquid-liquid flows due to large density difference in the gas-liquid flows. Further, for liquid-liquid flow in horizontal pipe, there is a difference in flow regime due to gravity.

It is well known that the interfacial forces existing at the interphase of liquid-liquid phases are responsible for mass and momentum exchange between the two phases. There are many drag expressions proposed to account for the effect of drag on fluid flow such as drag on rigid sphere [1], drag on single drop [2] and drag in the presence of adjacent drops [3]. To study the effect of lift force simulations have been carried out using several constant values for lift coefficient. Turbulent dispersion force is studied by varying the value of dispersion Prandtl number (DPN) in the simulation.

In the present paper, based on the two-fluid model (E-E), the three-dimensional fluid flow of oil-water system is simulated and the effect of interphase forces on the fluid flow in specially discussed. The simulations are carried out in a cylindrical horizontal pipe of 1.0 inch ID (internal diameter) for mixture velocity of 2.12 m/s and 46% input water. The experimental results of Soleimani [4] are used to make a comparison with the simulation.

2.0 MATHEMATICAL FORMULATION

The E-E model is based on interpenetrating continuum assumption. Here, all phases share the domain and interpenetrate as they move through it. Each phase is characterised by distinct fields of velocity and volume fraction. The governing equations are solved for each phase, considering the time averaged values. The governing equations proposed by Ishii [5] are more commonly used in fluid-fluid flows. The exchange between the phases is represented by source terms in conservation equations. The phases are assumed to share space in proportion to their volume fractions so as to satisfy the total continuity relation, *i.e*.

$$\alpha_d + \alpha_c = 1.0 \qquad (1)$$

Turbulence is assumed to be a property of continuous liquid phase. The dispersion of the phases due to turbulence is represented by introducing a diffusion term in the mass conservation equation.

In the present study dispersed phase drops are assumed to be of uniform size throughout. The sauter mean diameter (SMD) of the dispersed drops is calculated using correlation proposed by Angeli [6] Drop break-up and coalescence is not taken into account, *i.e*. drop-drop interactions are assumed to be negligible.

turbulence model is applied to describe the behaviour of the liquid phase. There is no thermal interaction and the flow is essentially Newtonian, incompressible and unsteady.

2.1 Governing Equations

Within the framework of the above assumptions, the governing equations may be written in the following forms:

2.1.1 Continuity Equation (mass conservation equation)

For any phase q, the mass conservation equation can be written as:

$$\nabla.(\alpha_q \rho_q U_q - D\nabla\alpha_q) = 0 \tag{2}$$

where the second term represents the phases diffusion term which accounts for the dispersion of the dispersed drops in continuous phase. The diffusion or dispersion coefficient is associated with the random motion associated with the phases.

2.1.2 Conservation of Momentum

The momentum conservation equation can be written as:

$$\nabla.\left[\underbrace{\alpha_q \rho_q U_q U_q}_{convection} - \underbrace{\alpha_q \Gamma_q U_q}_{diffusion}\right] = \underbrace{S_q + S^*_q}_{Source} \tag{3}$$

where is an exchange coefficient representing the effects of turbulent diffusion within the phases. The effective viscosity is given as:

$$\mu_{eff} = \mu_t + \mu_c \tag{4}$$

where the turbulent viscosity is given by

$$\mu_t = C\mu \rho_c k^2/\varepsilon \tag{5}$$

Turbulence in continuous phase is modeled in the same way as that of single phase flow using Equation (6) with transport equations for k and ε:

$$\frac{\partial}{\partial t}(\alpha_q \rho_q \varphi_q) + \nabla.(\alpha_q \rho_q \bar{U}_q \varphi_q) = \nabla.\left[\alpha_q \frac{\mu_{t,q}}{\partial_{pr_\varphi}} \nabla\varphi_q\right] + S_{\varphi q} \tag{6}$$

The term in Equation (6) is further expanded as follows:

$$S_{kq} = \alpha_q G_{k,q} - \alpha_q \rho_q \varepsilon_q + \alpha_q \rho_q \prod k_q \tag{7}$$

$$S_{\varepsilon q} = \alpha_q \frac{\varepsilon_q}{k_q}[C_{1\varepsilon} G_{k,q} - C_{2\varepsilon} \rho_q \varepsilon_q] + \alpha_q \rho_q \prod k_q \tag{8}$$

where $G_{k,q}$ is the generation of turbulent kinetic energy due to mean velocity gradient in phase $_q$, which is again calculated as:

$$G_k = \mu_{eff} \frac{\partial U_i}{\partial x_i}\left[\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right] \tag{9}$$

2.2 Interfacial Forces F_i

The drag expression for single rigid sphere is generally expressed by the correlation proposed by Schiller and Naumann [1]:

$$C_{D0} = \begin{cases} 24(1+0.15\text{Re}^{0.687})/\text{Re} & \text{Re} \leq 1000 \\ 0.44 & \text{Re} \leq 1000 \end{cases} \tag{10}$$

However, a liquid drop in another liquid medium does not behave like a rigid sphere [7]. Expressions to account for drag on a single drop have been proposed by many researchers [2, 8-10]. The general expression for drag on a single drop is given by Equation (11).

$$F_D = \frac{1}{8}\pi d^2 C_{D0} \rho_c V_s^2 \tag{11}$$

However, drops moving in the presence of adjacent drops experiences more drag because its motion is now impeded by other drops. The drag on drops moving with adjacent drops is given by Equation (12):

$$F_D = \frac{3}{4}\frac{C_{DM} \alpha_d \rho_c V_s^2}{d} \tag{12}$$

The new drag coefficient C_{DM} in Equation (12) accounts for reduced buoyancy and increased drag due to the presence of other drops. The expressions proposed for drag force which takes into account the presence of adjacent drops [2, 3, 11] are listed in Table 1.

Table 1: Drag expressions on drops which take into account the presence of adjacent drops [7]

Investigator (s)	*Proposed expression for drag coefficient (C_{DM}) in the presence of adjacent drops*
Hu and Kinter [8]	$\mu_m = \mu_c(1-\alpha_d)^{-2.5(\mu_d+0.4\mu_c)/(\mu_d+\mu_c)}$ and $Re_m = \frac{d_e V_s \rho_c}{\mu_m}$ $C_{DM} = \frac{24}{Re_m}$ } *Stokes regime* $C_{DM} = \frac{24}{Re_m}(1+0.1Re_m^{0.75})$ } *Undistorted particle regime* $f(\alpha_d) = \sqrt{1-\alpha_d}\,(\frac{\mu_c}{\mu_m})$ and $E(\alpha) = \left\{\frac{1+17.67[f(\alpha_d)]^{6/7}}{18.67 f(\alpha_d)}\right\}^2$ $C_{DM} = 0.45\,E(\alpha)$ } *Newton's regime* $C_{DM} = \frac{2}{3}\,Eo\,(E(\alpha))$ } *Distorted particle regime* $C_{DM} = \frac{8}{3}(1-\alpha_d)^2$ } *Chun turbulent flow regime*
Kumar and Hartland [3]	$C_{DM} = \left[0.53 + \frac{24}{Re}\right](1+4.56\alpha_d^{0.73})$
Rusche and Issa [11]	$C_{DM} = C_{D0} f(\alpha)$ *where* $f(\alpha) = exp(2.10\alpha_d) + \alpha_d^{0.249}$

The expression for slip (relative) velocity is given by:

$$V_S = \frac{4gd_e\Delta\rho(1-\alpha_d)}{3\rho_c C_{DM}} \tag{13}$$

Lift force in two-phase flows is primarily responsible for non-uniform radial distribution of the dispersed phase hold-up particularly in turbulent flows. Wall peaking, near wall peaking and coring profiles were experimentally observed in the experimental studies [4, 12]. The lift force used in CFD model assumes the following form, where the volume of the sphere is replaced by volume of the dispersed phase (α_d) since, the size and shape of the drops is assumed to be constant and non deformable.

$$\bar{F}_{lift} = -C_L\rho_q\alpha_d(\bar{v}_q - \bar{v}_p) \times (\nabla \times \bar{v}_q) \tag{14}$$

where C_L is referred to as lift coefficient in Equation (14). Lift coefficient is a single value which can be either positive or negative. It does not change with the local hydrodynamic condition or flow properties. There are no values of lift coefficients reported for dispersed liquid-liquid flows. Thus, in the present study simulations have been carried out using both positive and negative values (*i.e.* 0, -0.05, 0.5 and 1.0) based on the studies of Madhavan [7].

In turbulent flows, the dispersed entities are transported by turbulent eddies from region of high concentration to low concentration in the continuous phase which is termed as turbulent dispersion. It is strongly influenced by the fluctuating velocity field of the continuous phase which counteracts the lift force and homogenizes the flow in the domain. At high mixture velocity it was observed that the oil drops tend to move towards the core of the pipe due to the combined effect of inviscid lift and turbulent dispersion forces [4,12]. The drops tended to move to the core of the pipe in the region of low turbulence as turbulent dispersion is stronger near the wall. It was also reported that drops near the core region had the same diffusivity as of continuous phase. In the present study Simonin and Viollet [13,14] model is adopted, which uses the DPN to signify the turbulence dispersion.

2.3 Boundary Conditions

Figure 1 shows the computational domain used in this present study. It shows the hexahedral mesh with approximately 160,000 cells and the boundary conditions defined.

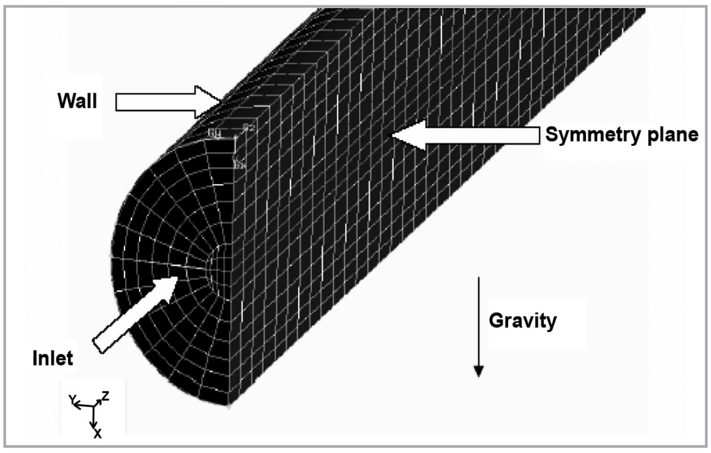

Figure 1: Computational domain showing boundary conditions

2.3.1 Inlet Conditions

Uniform inlet velocity is considered at the pipe inlet. Velocity of both the phases and volume fraction of the dispersed phase is given at the inlet.

$$U_x = 0 \qquad U_y = 0 \qquad U_z = 1 \tag{15}$$

Turbulence intensity (I) is calculated as:

$$I = 0.16(R_e)^{-1/8} \tag{16}$$

The turbulence parameters at the inlet are calculated as:

$$k = \frac{3}{2}\left[\frac{U_c}{\alpha_c} I\right] \text{ and } \varepsilon = C_\mu^{3/4}\frac{k^{3/2}}{l} \tag{17}$$

where l is the turbulent length scale. For fully developed flows $l = 0.07D$, where D is the pipe diameter.

2.3.2 Outlet Conditions

At the outlet of the pipe pressure outlet boundary condition is implemented. Atmospheric pressure is specified.

2.3.3 Wall Conditions

Wall boundary condition was used to bind the fluid and the solid region. '*No-slip*' boundary condition was imposed at the wall.

$$U_x = 0 \qquad U_y = 0 \qquad U_z = 0 \tag{18}$$

However, to determine the solution of the governing equations at the solid wall, where steep gradients in the flow variables occur, a log law wall function is employed to simulate the turbulence behaviour near the wall.

The whole computational domain is divided into viscosity affected region and fully turbulent region based on turbulence Reynolds number, Re_y, defined as:

$$\text{Rey} = \frac{\rho_c y_w \sqrt{k_c}}{\alpha_c} \tag{19}$$

where y_w is the normal distance from the wall. Standard wall functions, Y+ values in the range of 30-60 (closer to lower bound) are used. The Y+ value of 35 is used to calculate the first grid point from the wall. In viscosity affected region near the wall where turbulent Reynolds number (Re_y) is less than 200, about 8-12 cells of fine spacing are used to resolve the mean velocities and turbulent parameters.

2.3.4 Symmetry Conditions

The turbulent liquid-liquid flows are considered to be symmetric about the center plane. *Symmetry* boundary condition is used to reduce the computational cells to half thus reducing the computational time.

3.0 RESULTS AND DISCUSSION

A finite volume method CFD code FLUENT is employed to solve the governing equations numerically. The computational domain consists of approximately 160,000 hexagonal cells.

In the present study, the mathematical simulations are carried out for the following four cases:

(i) Case I: only effect of different drag force is taken into account as the interphase force to study the phase distribution.

(ii) Case II: using drag expression of case I, effect of lift coefficient is studied using range of positive and negative values.

(iii) Case III: effect of individual forces is studied on phase distribution.

3.1 Case I

Figure 2 shows the comparison of different drag models proposed in literature: drag on rigid sphere [1], drag on single drop [8], drag on single drop in dense dispersion [11] and drag on a single drop in the presence of adjacent drops [3].

The simulations are carried out at 46% water fraction and 2.12 m/s mixture velocity. The drag expression of Schiller and Naumann [1] shows stratification which water hold-up of 80% at the bottom and 20% at the top wall of the pipe. The drag expression for single drop shows comparatively better mixing. The drag expression of Rusche and Issa [11] shows slightly better results then drag expression of Kumar and Hartland [3]. However, drag expression proposed by Rusche and Issa [11] is valid only for dense dispersions and cannot be applied for dispersed phase volume fractions less then 30%, while drag expression proposed by Kumar and Hartland [3] is applicable for dispersed phase volume fractions greater then or equal to unity. Hence, in the present study the drag expression proposed by Kumar and Hartland [3] is used whose predictions lies between drag on a single drop and drag in dense dispersions to predict the phase distribution for dispersed flow in horizontal pipeline.

3.2 Case II

Here the simulations are carried out using three different values of lift coefficient, *i.e.* -0.05, 0 and 1, respectively in conjunction with drag expression proposed by Kumar and Hartland [3] with DPN of 7.5. As shown in Figure 3, for $C_L = 1$ and 0.5, the phase distribution profiles shows near wall peaking at top wall while at the bottom of the pipe wall the phase is more homogeneously mixed for $C_L = 1$. For $C_L = -0.05$, the phase distribution profile shows wall peaking with considerable amount of water hold-up at top and bottom walls. The phase distribution profile for

$C_L = -0.05$ is similar to that observed by other researchers [4, 12].

3.3 Case III

Figure 4 shows the effect of individual inperphase force on phase distribution profile.

The results in Figure 4 clearly show that there is no considerable effect of lift and turbulent dispersion forces on phase distribution. There is no significant difference observed between the phase distribution profiles using only drag force and with all three interphase forces. When only turbulent dispersion force is used (DPN = 7.5) the flow is almost stratified while with only lift as the interphase force, the phase distribution profile at the bottom of the pipe shows separation while near wall peak is seen at the top wall. The possible reason may be due to the use of constant value of lift coefficient and turbulent dispersion force in the present study.

Figure 5 shows the comparison of the predicted phase distribution profile with experimental data of Soleimani [4].

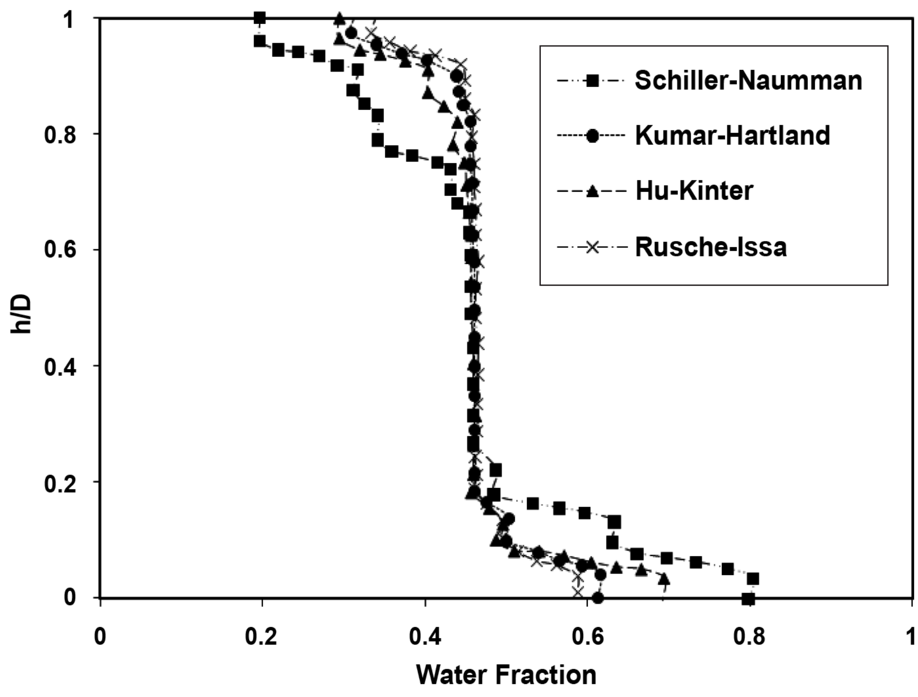

Figure 2: Comparison of drag models to study phase distribution profile at pipe cross section

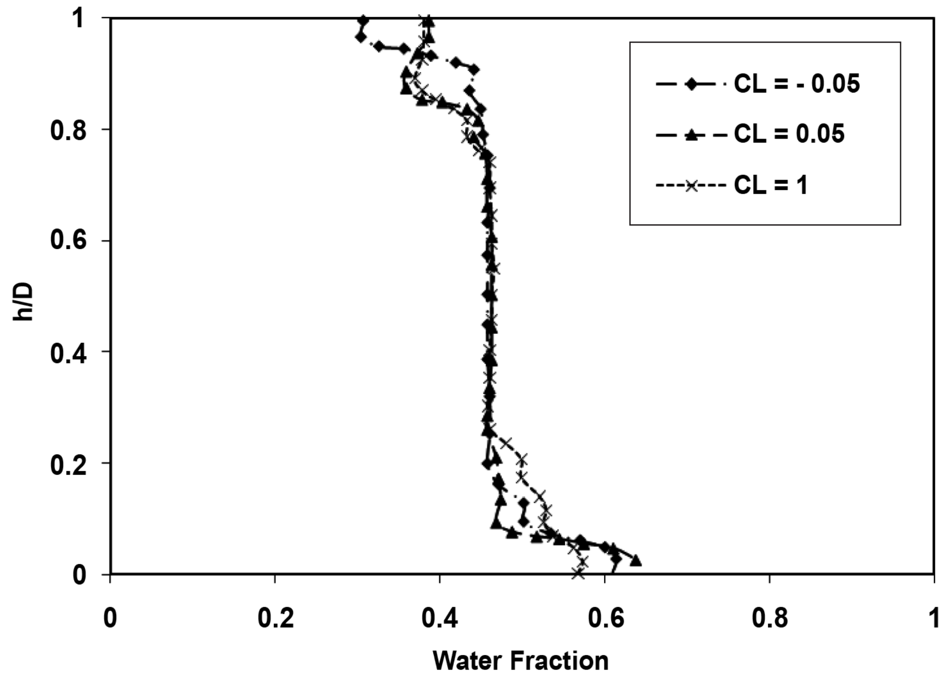

Figure 3: Phase distribution profile for different lift coefficient (C_L)

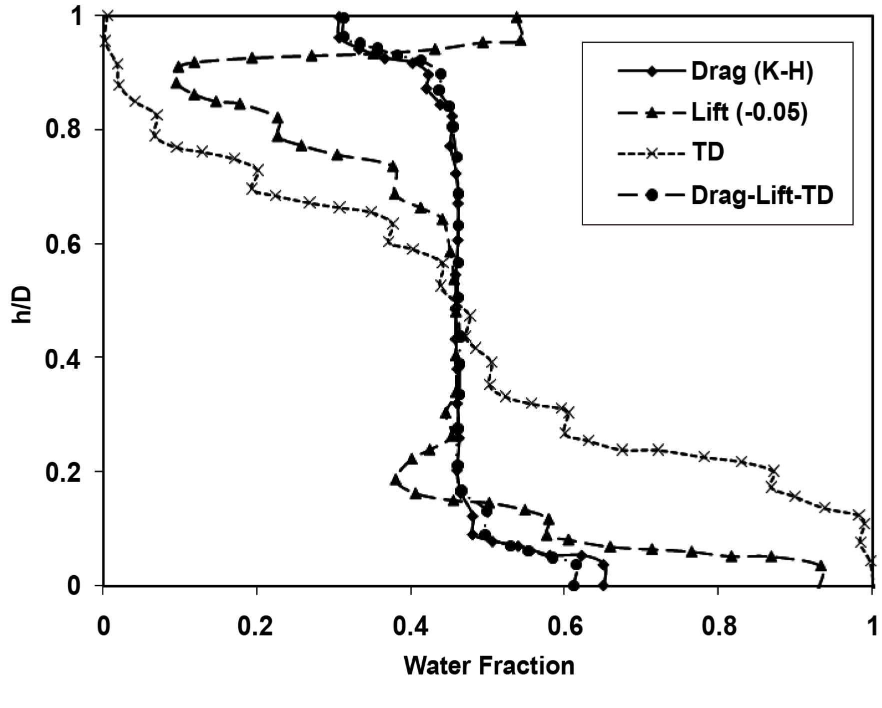

Figure 4: Effect of individual interphase forces on phase distribution profiles

It is observed that the phase distribution across the pipe cross section is in good agreement with the experimental data of Soleimani [4]. Effect of drag, lift and turbulent dispersion forces have been taken into account. All three forces need to be included in order to predict accurately the phase distribution in a dispersed liquid-liquid flow regime.

4.0 CONCLUSION

The effect of interphase forces on dispersion of oil-water mixture in a horizontal pipe is studied using CFD. The phase distribution profiles at pipe cross section are investigated. The results showed that drag is the major interphase force responsible in the phase distribution. However, for accurate prediction, the effect of lift and turbulent dispersion forces should be included.

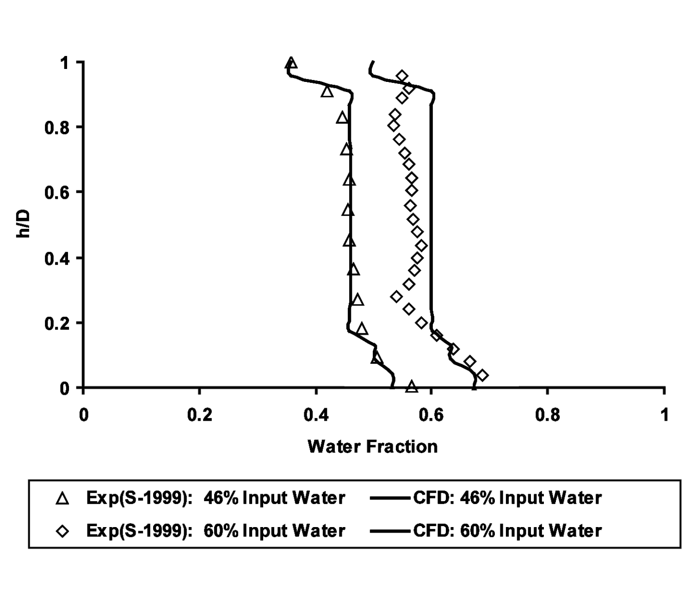

Figure 5: Comparison of vertical phase distribution profile at 46% and 60% input water and 3.0 m/s mixture velocity for data set of Soleimani [4]

5.0 ACKNOWLEDGEMENT

Rahsmi would like to acknowledge Universiti Putra Malaysia for research grant and also to the research team members for their support and guidance throughout this study. ■

NOTATION

C	constant
C_{D0}	Drag coefficient for rigid sphere
C_{DM}	drag coefficient in presence of adjacent drop
C_L	Lift coefficient
D	diffusion
d	drop diameter
F	force
G	rate of generation
g	Acceleration due to gravity
I	turbulence intensity
k	turbulent kinetic energy
l	turbulent length scale
R_e	Reynolds number
S	source term
U,V	velocity
Greek letters	
α	volume fraction
μ	viscosity
ε	turbulent dissipation rate
ρ	density
Γ	exchange coefficient
$\prod$	time
Subscript	
c, q	continuous phase
d, p	dispersed phase
s	slip
t	turbulent

REFERENCES

[1] Schiller, L., and Naumann, Z.Z. (1933). Uber Die Grundlegenden Berechungen Bei Der Schwerkraftbereitung, Z. Ver. Deutsch. Ing 77, 318-320.

[2] Ishii, M., and Zuber, N. (1979). Drag Coefficient and Relative Velocity in Bubbly, Droplet or Particulate Flows. American Institute of Chemical Engineers Journal 25, 843-855.

[3] Kumar, A., and Hartland, S. (1985). Gravity Settling in Liquid/Liquid Dispersions. Canadian Journal of Chemical Engineering 63, 368-376.

[4] Soleimani, A. (1999). Phase Distribution and Associated Phenomena in Oil-Water Flows in Horizontal Tubes. PhD Thesis, Imperial College, London.

[5] Ishii, M. (1975). Thermo-Fluid Dynamic Theory of Two-Phase Flow, Direction des Etudes et Recherches d'Electricité de France, Eyrolles, Paris, France.

[6] Angeli, P. (1996). Liquid-Liquid Dispersed Flows in Horizontal Pipes. PhD Thesis, Imperial College, University of London.

[7] Madhavan, S. (2005). CFD Simulations of Immiscible Liquid Dispersions. M.Sc Thesis, Dalhousie University, Canada.

[8] Hu, S., and Kintner, R. C. (1955). The fall of Single Liquid Drops through Water. American Institute of Chemical Engineers Journal., 1, 42-48.

[9] Klee, A. J., and Treybal, R. E. (1956). Rate of Rise and fall of Liquid Drops. American Institute of Chemical Engineers Journal 2, 444-447.

[10] Grace, J. R., Wairegi, T., and Nguyen, T. H. (1976). Shapes and Velocities of Single Drops and Bubbles Moving Freely Through Immiscible Liquids. Transaction of the Institution of Chemical Engineers 54,167-173.

[11] Rusche, H., and Issa, I. (2000). The Effect on Voidage on the Drag Force on Particles in Dispersed Two-Phase Flow. Japanese-European Two-Phase Flow Meeting, Tsukuba, Japan.

[12] Hussain, S. A. (2004). Experimental and Computational Studies of Liquid-Liquid Dispersed Flows. PhD Thesis, Imperial College, London.

[13] Simonin, O., and Viollet, P. (1990a). Predictions of an Oxygen Droplet Pulverization in a Compressible Subsonic Coflowing Hydrogen Flow. Numerical Methods for Multiphase Flows FED 91, 65-82.

[14] Simonin, O., and Viollet, P. (1990b). Modeling of Turbulent Two-Phase Jets Loaded With Discrete Particles. Proceedings of International Seminar on Phase Interphase Phenomena in Two-Phase Flow, Dubrovnik, ICHMT Belgrad.