CFD MODEL FOR DETERMINING LOCAL PHASE FRACTION OIL-WATER DISPERSION IN TURBULENT FLOW

Siti Aslina Hussain[1] and Soo Mee Khuan[1]*

[1]Department of Chemical and Environment Engineering, Faculty of Engineering, Universiti Putra Malaysia, Serdang, 43400, UPM, Selangor, Malaysia
Tel No.: +60389466292; Email Add.: aslina@eng.upm.edu.my

Abstract

Oil-water holdup for turbulent flow are commonly available, however, the common models are not easily implemented for fast computation of dispersion oil-water system in horizontal tube due to gravity-induced. The oil-water dispersion was studied numerically in a horizontal pipe had been varied out using advanced computational fluid dynamics, CFD software (CFX11.0 code). The proposed model includes the choice of parameters in CFD for a dispersed flow with 25.4 mm inner diameter and 9.7 m length in horizontal tube model to obtain the best agreement with the previous research work. In this work, input water fraction, 40, 46 and 60% of water controlling the performance of flow characteristics at mixture velocity of 1.8 and 2.76 m/s. With the advanced numerical simulation that can provide a better understanding of the module performance, especially for turbulent flow system and it is found to be fairly well-dispersed in the two-phase system. Hence, the CFD can be used to better apprehend fluid flows in complex geometries and to test the influence of the type of model and parameters such as Eulerian-Eulerian model with k-turbulence model can be successfully implemented to simulate the liquid-liquid flow in horizontal tube.

Introduction

Multiphase flow is the simultaneous flow of two or more phases in direct contact in a given system. It is important in many areas of chemical and process engineering and in the petroleum industry, e.g. in production wells and in subsea pipelines. The behavior of the flow will depend on the properties of the constituents, the flow rates and the geometry of the system. There are four combinations of two-phase flows namely, gas-liquid, gas-solid, liquid-liquid and solid-liquid. Liquid-liquid flows, the subject of the present project are extremely important particularly in two-phase flow applications in horizontal pipes, for instance in the oil industry.

In liquid-liquid flow systems, it is important to understand the nature of the interactions between the phases and to observe the ways in which the phases are distributed over the cross section of the pipe (i.e. the “flow regime” or “flow pattern”). In design, it is necessary to be in a position to predict pressure drop which, usually, will depend not only on the flow pattern, but also on the superficial velocities of the phases and the distribution of the fraction occupied by each phase over the cross section of the pipe. The mean in-situ volume fraction will not normally be the same as the input volume fraction.

A number of recent studies on oil-water dispersions have focused on horizontal pipelines and, in particular, on the flow distribution in the system. Arirachakaran *et al.,* 1989, Angeli, 1996, 2000, Soleimani, 1999 and Siti Aslina, 2004 found that dispersed flow for oil-water systems in horizontal pipes occurs when the liquid-liquid mixture is moving at high velocity. However, not all research studies done based on numerically work, where to get a better understanding on the predicted flow distribution in the system. In the numerical work usually researchers used to solve the mathematical equations known as Computational Fluid Dynamics (CFD). CFD has been widely used as a tool in industry to model flows at various levels of geometric complexity. Essentially it is used to solve the coupled mass, energy and momentum equations. This can be done by representing the geometry as a mesh of cells and solving coupled equations for each cell. In addition to its now-routine use in predicting single phase turbulent flows, CFD is increasingly used in predicting dispersed multiphase systems.

In 1999, Soleimani obtained that the model developed gave a fairly agreement with the experimental data and the flow distribution represent a homogeneous distribution at high mixture velocity, however, there are several limitations in the model. In 2004, Siti Aslina found that the flow pattern progression from partially separated to fully mixed conditions as the velocity is increased from 1.8 to 2.76 m/s. However, in the same year, she found that the model developed indicated a tendency to separate. This may due to another mechanism for mixing in predicted results perhaps Kelvin-Helmholtz instability in turbulent flow Siti Aslina (2004, 2007).

In general, CFD is a useful tool in the study of the multiphase processes studies and has been used extensively in the present work. The aim has been to compare the previous numerical results which is used CFX 4.4 code obtained by Siti Aslina, 2004 for phase distributions in developing liquid-liquid dispersed flows with that predicted from a current commercial CFD code (CFX 11.0). The used of Eulerian-Eulerian model with k-turbulence in the current model. All in all, comprehensive understanding of the physics of fluid flows and the fundamentals of the numerical algorithms are the key elements in CFD to produce a consistent, stable, convergent, conservation, boundedness, realizable and accurate result.

Methodology

Computational Fluid Dynamics (CFD) is the science of predicting fluid flow, heat and mass transfer, chemical reactions, and related phenomena by solving numerically the set of governing mathematical equations, for example conservation of mass, momentum, energy, species, etc. The results of CFD analyses are relevant in conceptual studies of new designs, detailed product development, troubleshooting and redesign. Apart of that, CFD analysis complements testing and experimentation that reduces the total effort required in the experiment design and data acquisition. Figure 1 overview the CFX 11.0 with 5 modules.

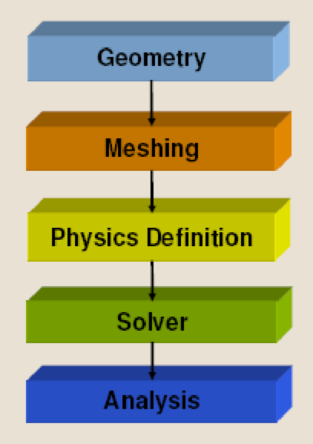


Figure 1. Overview the CFX11.0

In computational modelling of dispersed two-phase flow, the flow of the individual phases is represented as two separate interacting fields, one of which is dispersed and the other continuous. The continuous phase occupies a connected region of space and the disperse phase occupies disconnected regions of space. Dispersed two-phase flow refers to the flow of a dispersed phase, such as droplets, in a continuous phase such as a liquid. The motion of the dispersed phase is dependant on the forces exerted as it by the continuous phase for examples drag forces and non-drag forces (including lift, virtual mass and turbulent dispersion forces).

Physical Property Data

Water and oil (EXXSOL D-80) were used as the input substances into the horizontal pipe. The code requires a range of physical properties and the ones used are given in Table 1.

Table 1. Physical property data of oil

Parameters	Values	Reference
Thermal Expansivity [K^{-1}]	990×10^{-6}	*http://www.engineeringtoolbox.com*
Dynamic Viscosity [cP]	1.6	Siti Aslina, 2004, 2007
Density [kg m^{-3}]	801	Siti Aslina, 2004, 2007
Molar Mass [kg $kmol^{-1}$]	170.88	Perry Robert H., 1997
Specific Heat Capacity [cal g^{-1} K^{-1}]	0.48	*http://www.engineeringtoolbox.com*
Thermal Conductivity [W m^{-1} K^{-1}]	0.145	*http://www.engineeringtoolbox.com*

Local phase fraction

Clearly input water fraction is a very important variable. At the lowest water fraction studied (40%) the water would be expected to be dispersed in the oil and at the highest (60%) the oil would be expected to be dispersed in the water with varied total flow velocities. The gravitation direction is set up downward which is at the –y direction. All data and measurements were taken based on Siti Aslina, 2004. The current numerical matrix on water-oil phase fraction is summarised as below in Table 2.

Table 2. Numerical matrix on water-oil phase fraction

Parameters	CFD Model			
	1	**2**	**3**	**4**
Water Volume Fraction	0.60	0.46	0.40	0.60
Oil Volume Fraction	0.40	0.54	0.60	0.40
Mixture Velocity [m s^{-1}]	1.80	1.80	1.80	2.76

Adjustable Parameter in Parameter in CFX 11.0

Since it would be unrealistic to expect CFD codes to predict complex system like the one considered here, the normal practice is for the user to have available a range of adjustable parameters which can be modified to bring the results closer to the previous research work values. In this sense, CFD of complex flows is more realistic to the liquid-liquid flow system. Table 3 gives a listing of the adjustable parameters, these default value and the values actually used in the present calculations. The effects of the adjustments were investigated by examining their influence on the predicted vertical distributions of water fraction.

Table 3. Selection of CFX 11.0 adjustable parameters

Parameters	Default Values	Specified Values
Body Spacing [mm]	490	5
Face Spacing [mm]	25	5
Angle Resolution [˚]	30	30
Number of Inflated Layers	5	5
Expansion Factor	1.2	1.1
Minimum Internal Angle [˚]	2.5	30
Maximum Internal Angle [˚]	10.0	10.0
Inflation Option	Total Thickness	First Layer Thickness
First Prism Height [mm]	49	1
Breakup Coefficient	1.0	1.0
Buoyancy Coalescence Coefficient	1.0	1.0
Turbulence Coalescence Coefficient	1.0	1.0
Surface Tension Coefficient [N m^{-1}]	0.017	0.017
Lift Coefficient	0.5	0.5
Turbulence Dispersion Coefficient	1.0	1.0
Virtual Mass Coefficient	0.5	0.5

In this case, if the water was assumed to be the continuous fluid which formed a continuous connected region; while oil was assumed to be the polydispersed fluid which was present in the discrete regions which were not connected. The above conditions fulfil the criteria of an Eulerian-Eulerian multiphase model. CFX 11.0 code offers the option to choose between a numbers of different models to represent the continuous phase turbulence. The standard k-ε model of turbulence was chosen as it has proven to be stable, has a well established regime of predictive capability and offers a good compromise in terms of accuracy and robustness (Bode, 1994; Yang G. *et. al.*, 2007). In the k-ε model, both k and ε which are turbulent kinetic energy and turbulent dissipation rate respectively must be specified. The turbulent kinetic energy was related to the turbulence intensity by the equation:

$$k = (uI)^2 \tag{1}$$

where I is the intensity of turbulence. The turbulence intensity was estimated from the following equation for turbulence in pipe flow given by Langrish and Zbicinski (1994):

$$I = 0.2\,\mathrm{Re}^{-1/8} \tag{2}$$

Here Re is the Reynolds number of the inlet pipe flow. For a Reynolds number of 35,776 corresponding to the velocity in the pipe of 1.80 m s^{-1} and the inlet internal diameter of 25.4 mm, this equation predicts that the turbulence intensity takes the value of 0.0539. The average value turbulence kinetic energy calculated from equation (1) and (2) is 9.4234 X 10^{-3} m^2 s^{-2}.

The turbulent dissipation rate was not measured directly but was estimated from a relationship between the length scale of turbulence and the turbulence kinetic energy given by Langrish and Zbicinski (1994):

$$\varepsilon = \frac{C_\mu^{\,3/4} k^{3/2}}{\ell} \tag{3}$$

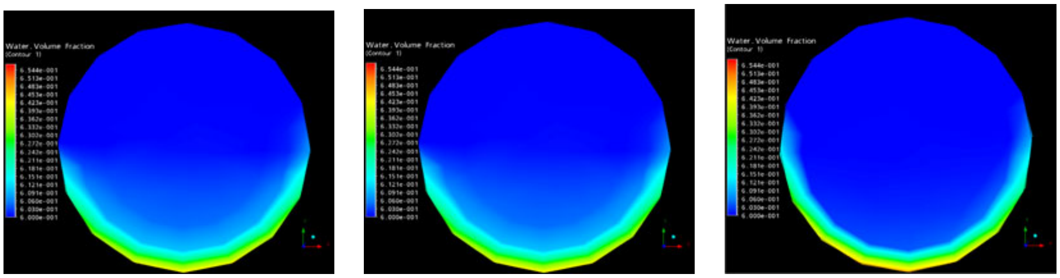
where ℓ is the length scale of turbulence and C_μ is the constant. The length scale of turbulence was assumed to be the diameter of the inlet internal diameter which is 25.4 mm. On this basis, an estimate of 5.9178 X 10^{-3} m^2 s^{-3} is obtained for the turbulent dissipation rate across the inlet. Table 3.3.3 illustrates the calculated values for both k and ε using equation (3), (4) and (7).

Table 4. Calculated values for k and ε using equation (1), (2) and (3).

Parameters	**CFD Model**			
	1	**2**	**3**	**4**
Water Volume Fraction	0.60	0.46	0.40	0.60
Oil Volume Fraction	0.40	0.54	0.60	0.40
Mixture Velocity [m s^{-1}]	1.80	1.80	1.80	2.76
Fractional Intensity, I	0.0539	0.0547	0.0550	0.0511
Turbulent kinetic energy, k [m^2 s^{-2}]	9.4234 X 10^{-3}	9.6901 X 10^{-3}	9.8026 X 10^{-3}	1.9910 X 10^{-2}
Turbulent dissipation rate, ε [m^2 s^{-3}]	5.9178 X 10^{-3}	6.1708 X 10^{-3}	6.2786 X 10^{-3}	1.8174 X 10^{-2}

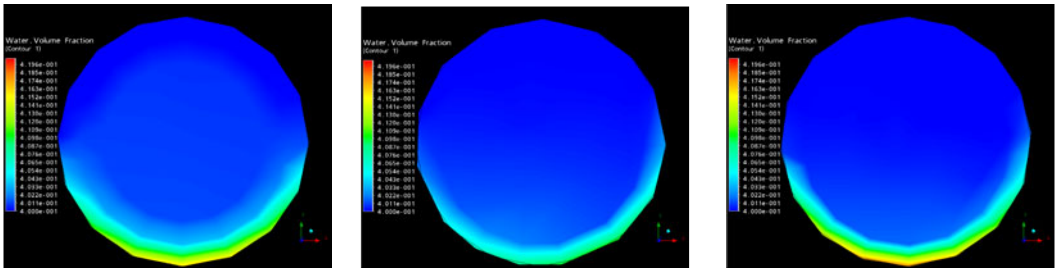
Results

The effect of the inlet water fraction



(a) 1.00 m from pipe inlet (b) 5.85 m from pipe inlet (c) 7.72 m from pipe inlet

Figure 2. Effect of position on calculated cross-sectional phase distributions for a 60% input water fraction and a mixture velocity of 1.8 m/s.



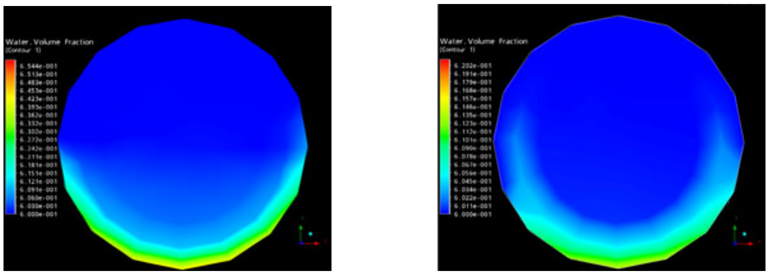
(a) 1.00 m from pipe inlet (b) 5.85 m from pipe inlet (c) 7.72 m from pipe inlet

Figure 3. Effect of position on calculated cross-sectional phase distributions for a 40% input water fraction and a mixture velocity of 1.8 m/s.

At the inlet, the oil is introduced at the top of the channel and the water at the bottom. The phase distribution observed at 1.0 m for a mixture velocity of 1.8 m/s may strongly reflect this initial distribution with oil-rich and water-rich zones being mixing throughout the channel. At 60% input water, the oil-in-water dispersion would be expected and the water phase fraction varying within the range 0.6 to 0.65 along the channel. At 40% input water, the water-in-oil dispersion would be expected with values of the water phase fraction within the range of 0.4 and 0.42 along the channel. Both input waters indicate that the flow in the system is fairly uniform distributed across the cross sectional of the pipe along the channel. This is because turbulence has occurred in the pipe and mixing the mixture flows towards to stability along the channel. Presumably, oil and water droplets were well-distributed in the water-in-oil or oil-in-water dispersion as depicted in Figure 2 and 3.

The effect of the mixture velocity

By comparing the water volume fraction contour plot at 1.0 m from the pipe inlet, the water volume fraction varies from 0.6000 to 0.6408 for a mixture velocity of 1.80 m/s while the water volume fraction varies from 0.60 to 0.61 for a mixture velocity of 2.76 m/s. Both mixture velocities give a small range of deviation that indicates a perfect mixing is achieved due to highly turbulent flow. The oil and water are uniformly well-mixed with well-dispersed distributions. Figure 4 illustrates the water volume fraction contour plot produced by CFD calculations of stationary turbulent flow with 60 % water input at mixture velocity (a) 1.8 m/s and (b) 2.76 m/s..



(a) Mixture velocity of 1.80 m s^{-1} (b) Mixture velocity of 2.76 m s^{-1}

Figure 4. Comparison of water volume fraction contour plot produced by CFD calculations of stationary turbulent flow with 60 % water input with different mixture velocity at 1.0 m from pipe inlet.

Uniform initial distributions of the dispersed phase

Here, the concentration of the dispersed phase was assumed constant and equal across the inlet plane, its value being equal to the input volume fraction. The drop size was also assumed constant across the inlet plane with value from Siti Aslina, 2004 with 1.42 X 10^{-3} m for maximum and 1 X 10^{-5} m for minimum of drops. Figure 5 illustrates the height ratio plotted against water volume fraction for various distances from the pipe inlet.

The results demonstrate the phase along the channel uniformly distributed across cross sectional of the pipe at 0.6. However, s mall amount of water droplets appear at the bottom due to heavy phase which is clearly seen by plotting the vertical distributions of chordal mean water fraction as in Figure 5. This indicates that the water volume fraction at 7.72 m has more drops at the bottom of the pipe compared to the top (Rashmi *et. al.*, 2006). As a conclusion, it can be said that the oil-water system are uniformly distributed due to the used of Eulerian-Eulerian model with k-turbulence model and adjustable parameter used in the CFX 11on the achievement of small variations in the result.

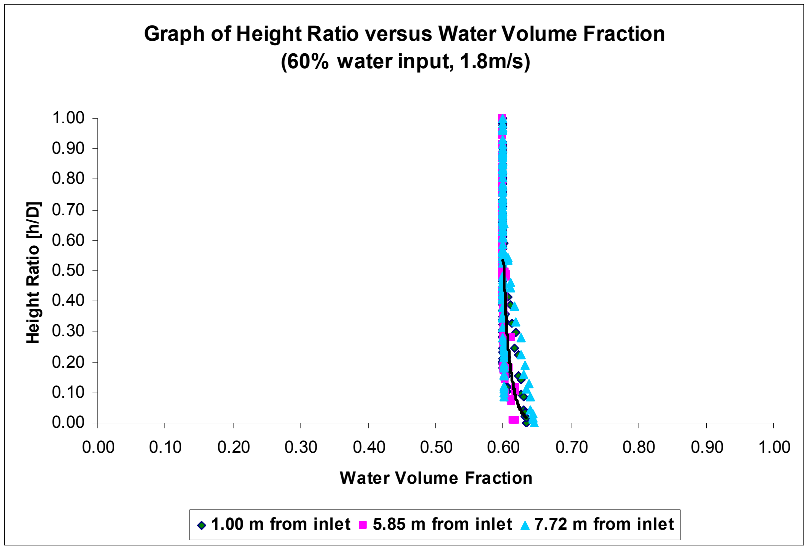


Figure 5. Comparison between height ratio and the water volume fraction for 60 % water input at various distances from the pipe inlet.

Comparison of local volume fraction results with previous studies

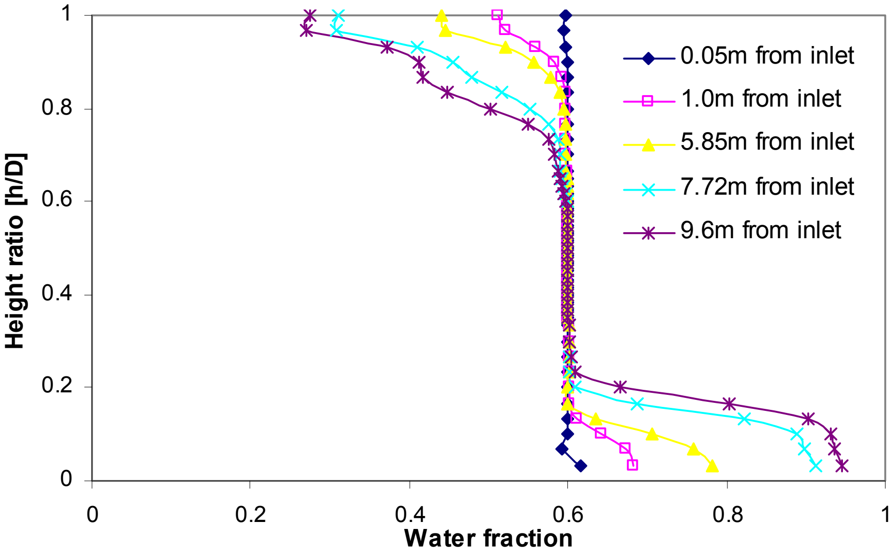


Figure 6. Vertical distribution of chordal mean water fraction as a function of position for 60% of input water fraction and a mixture velocity of 1.8 m/s (Siti Aslina, 2004).

Measurements of the vertical distributions of chordal mean void fraction were made by Siti Aslina, 2004 for conditions similar to those in the present numerical work. Comparisons between the present data and that reported by Siti Aslina, 2004 are shown in Figure 5 and 6. For the velocity 1.8m/s, there is a greater separation with lower water concentration at the top of the pipe and higher at the bottom. However, the current result shows that a great phases distribution uniformly across the cross section of the pipe along the channel as shown in Figures 5.

This separation is contrary to the current numerical findings. This is probably for by the fact that Siti Aslina used an old version of CFD that incomplete with liquid-liquid system features while the latest version has an advanced tool especially for liquid-liquid flow system. Apart of that the used of Eulerian-eulerian model with k-e turbulence which is highly tendency toward to well distribute phase fraction across cross sectional of the pipe. Also, the adjustable parameters were set in CFX 11.0 to somewhat favour mixing.

This tendency is affected by the action of turbulent eddies in the continuous phase which act towards making uniform the distribution of the dispersed phase due to turbulent diffusion. The actual distribution is a manifestation of the balance between gravity-induced separation and turbulence-induced mixing.

The mixing processes assumed in CFX 11.0 are those of turbulent transport of the droplets. It is possible that additional mechanisms such as Kelvin-Helmholtz instability may have a significant role in promoting mixing. This is consistent with some unpublished results presented in a lecture at Isaac Newton institute by Ferguson in 1999 Ferguson carried out Large Eddy Simulation calculations on stratified liquid-liquid flows and showed the existence of large (probably Kelvin Helmholtz) distribution which strongly promoted mixing.

Conclusion

Numerical study of oil-water flow system in a horizontal pipe had been analyzed in this study using ANSYS CFD (CFX 11.0 code). A horizontal pipe with 25.4 mm internal diameter and 9.7m length was designed for the liquid-liquid flow. It was developed in order to analyze the flow pattern and phase distribution of oil-water system in the horizontal pipe. The pipe was meshed into 227,820 elements with 164,306 tetrahedrons, 58,333 of prisms and 5181 of pyramids.

The present results have been compared with the previous numerical research results of Siti Aslina (2004, 2007). Some conclusions were made between the present model developed and Siti Aslina results. It seem that the oil and water distribution in the horizontal pipe is found to be fairly uniform and well-dispersed in the system for the present developed model compared to the previous model developed progressive separated with more water concentration at he bottom and highly oil concentration at the top of the pipe. Although the present formulation is rather complex and demands much computational resources and time due to the nature of the turbulent model and the fine grids required for its implementation, it does appear to demonstrate that the CFD technique can be successfully applied to this turbulent liquid-liquid flow.

The conclusion drawn from the present study is that the multiphase Eulerian-Eulerian model with k-ε turbulence model can be successfully applied to simulate the liquid-liquid flow in horizontal pipelines. The flow is considered to be homogeneous for the present study where the polydispersed phase is well-dispersed in the oil-water system. The third conclusion were used of MUSIG algorithm with non-drag forces (turbulent dispersion force, lift force and virtual force) included in the code to somewhat favour the mixing where MUSIG algorithm was developed to handle dispersed multiphase flows in which the dispersed phase has a large variation in size. Despite the selection of input parameters which also favour mixing.

Aknowledgement

We would like to express our gratitude to Universiti Putra Malaysia for funding this work.

References

1. Angeli. P (1996), "Liquid-liquid in Horizontal Pipes", *Ph.D thesis*, Imperial College, University of London, UK.
2. Arirachakaran, S, Oglesby, K.D, Malinowsky, M.S, Shoham, O and Brill, J.P (1989), "An Analysis of Oil/Water Flow Phenomena in Horizontal Pipes", SPE paper 18836, *SPE Prod. Operating Symposium*, pp. 155-167, Oklahoma, March 13-14.
3. Bode Dr. J. (1994), "Computational Fluid Dynamics Applications in the Chemical Industry", *Computers Chem. Engineering*, Vol. 18, pp 82-90.
4. Perry Robert H. and Green Don W. (2007), "Perry's Chemical Engineers' Handbook" 7th Edition, Mc-Graw Hill.
5. Langrish T. A. G. and Zbicinski I.(1994), "The Effects of Air Inlet Geometry and Spray Cone Angle on the wall Deposition Rate Spray Dryers", *Trans IChemE*, Vol. 72, Part A, May 1994.
6. Rashmi W, Siti Aslina H. and Thomas S. Y. Choong (2006), "CFD for Liquid-Liquid Dispersed Flows in Horizontal Pipe", *Seminar on Engineering and Technology*, Palm Garden Hotel, Putra Jaya, Malaysia, 4-5 September 2006.
7. Soleimani, A. (1999), "Phase Distribution and Associated Phenomena in Oil-Water Flows in Horizontal Tubes", *PhD Thesis*, Imperial College, University of London, UK.
8. Siti Aslina H. (2004), "Experimental and Computational Studies of Liquid-Liquid Dispersed Flows", *PhD. Thesis*, Imperial College, London.
9. Siti Aslina H., Siti Mazlina M. K., Wan Hassan M. J., Xiao Y. X. and Hewitt G. F. (2007), "Simulation on Oil-Water Horizontal Flows: MUSIG Model", *World Engineering Congress 2007*, Penang, Malaysia, 5-9 August 2007.
10. Yang G., Enqvist Y., Qu H., Louchi-Kultanen, Kallas J., Wanga J. (2007), "CFD Modeling of Solid-Liquid Suspension Flow in a Horizontal Pipe", *Proceedings of European Congress of Chemical Engineering* (ECCE-6), pp. 56-62.
11. http://www.ansys.com, *ANSYS Release 11.0 Documentation for ANSYS Workbench.* [Last accessed on 1st February 2008].
12. http://www.engineeringtoolbox.com/thermal-conductivity-liquids-d_1260.html, *Thermal Conductivity Common Liquids.* [Last accessed on 24th January 2008].