

SIMULATION ON OIL-WATER HORIZONTAL FLOWS: MUSIG MODEL

Siti Aslina Hussain^{1,*}, Siti Mazlina Mustapa Kamal¹, Wan Hassan Mohd Jamil²,
Xiao Yu Xu³, Geoffrey F. Hewitt³

¹Dept. of Chemical and Environmental Engineering, Faculty of Engineering, Universiti Putra Malaysia,

²Geowell. Sdn. Bhd. 151 Jalan Aminuddin Baki, TTDI 6000 Kuala Lumpur

³ Dept. of Chemical Engineering, Imperial College London, South Kensington Campus, United Kingdom

*Email: aslina@eng.upm.edu.my

ABSTRACT

*The most advanced of the commercial codes namely CFX 4.4 to predict the flow behavior from experimental data was used. The mathematical models adopted, which include the inter-phase drag, non-drag forces and **M**ultiple-**S**ize-**G**roup (**MUSIG**) model which was designed to handle dispersed multiphase flows in which the dispersed phase has a large variation in size. A CFD model was set up for liquid-liquid flow in a horizontal cylindrical tube with 25.4 mm ID and 9.7 m in length. It was developed to compute the spatial phase distribution of the oil and water in dispersed flow. The tube was divided into 200 axial cells and a grid with a total of 120,000 hexahedral cells. As a result, the CFD model which compares the predicted and actual evolution of the flow. The computations indicate a tendency to separate whereas the experiments indicate a tendency to mix*

Keywords: Dispersed flow, inter-phase drag, non-drag forces, Multiple-Size-Group, mixing

INTRODUCTION

Computational Fluid Dynamics (CFD) has been widely used as a tool in industry to model flows at various levels of geometric complexity for example for multiphase flow which 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.

Essentially CFD is used to solve the coupled mass, energy and momentum equations [5,6,7,8]. 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 [6]. At its simplest level, the applications of CFD to such systems is made using an assumption that the dispersed phase elements (bubbles, drops, particles) have a fixed and uniform size and shape (usually spherical)[7]. Obviously, this is an inadequate representation of actual fluid/fluid systems where bubbles and drops exist in a wide range of sizes.

Population balance equations have, however, been used successfully to calculate the distribution of phase element (bubble or drop) size between a number of size classes. The birth and death of bubbles or drops within each class occurs due to break-up and coalescence. Break-up and coalescence rate models have been developed from turbulence and statistical analysis of both processes. An example of such a model is the MUSIG model which was developed to incorporate population balance modelling into the commercial CFD code (CFX 4.4) to allow improved modelling of dispersed two-phase flow [8,9,15]. This allows the mono-dispersed assumption to be relaxed and the use of local averaged diameters to solve the momentum equation. In general, CFD is a useful tool in the study of the multiphase processes and has been used extensively in the present work. The aim has been to compare the experimental data obtained for phase distributions in developing liquid-liquid dispersed flows with that predicted from a commercial CFD code (CFX 4.4) which incorporates population balance modelling. The CFX code is supplied by ANSYS CFX Ltd.

COMPUTATIONAL FLUID DYNAMICS - CFX-4.4 CODES

CFX 4.4 is a CFD software developed by AEA Technology plc (and now owned and marketed by Ansys CFX Ltd) for the prediction of laminar and turbulent flow and heat transfer. It is a multiblock code which means that the geometry is made up of multiple deformed blocks which allows the modeling of more complex geometries. Each block can be transformed to a uniform grid without tearing or folding, using body fitted coordinates. This process can be controlled by the command language in the command file. In the pre-processors, CFX-Meshbuild creates multi-block meshes for use by the solver. Blocks are defined by their vertices and edges. The edges are subdivided before the grid generation is performed and patches are created to specify the location of the boundary conditions to flow solvers. CFX-Meshimport acts as an interface to convert unstructured, finite-element meshes into a form suitable for the solver and also can be used on the existing CFX 4.4 geometry files to optimize the block structure [7,8]. Plots of grids, velocity vectors, and shaded contours can be displayed using the visualisation package known as CFX-View [8]. CFX-Linegraph may be used to display graphs from columns of data and to carry out dynamic line graphing of residuals and monitoring points during a run of the solver [9,10]. CFX 4.4 consists of three modules which are Pre-Processing, Solver and Post-processing

In computational (CFD) modeling 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 dispersed 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. There are two basic multiphase flow models available in CFX 4.4, the multi-fluid model and homogeneous model; the latter (which allows detailed prediction of interfacial structure in simple cases) was not used in this study. Obviously, the motion of the dispersed phase is dependant on the forces exerted as it by the continuous phase. These forces comprise drag forces and non-drag forces (including lift, virtual mass and turbulent dispersion forces). It is also necessary to specify the identity of the continuous phase. In its simplest form, the CFX 4.4 code model for dispersed flow assumes a constant dispersed element size (i.e. drop size in the present study). However, the code can also perform calculations in which the drop size is calculated using a population balance model (MUSIG). **MUSIG** stands for **M**Ultiple-**S**ize-**G**roup [9,15]. The **MUSIG** model was developed to handle dispersed multiphase flows in which the dispersed phase has a large variation in size. **MUSIG** provides a framework in which a population balances method together with break-up [16] and coalescence [17] models can be incorporated into three-dimensional CFD calculations. In general, droplets in the dispersed phase can be divided into several size groups and each of these size groups can be treated as a separate phase in the multiphase flow calculation.

RESULTS

COMPUTATION MESH AND BOUNDARY CONDITION

The calculations using CFX 4.4 were aimed at representing a 9.7 m length of the 25.4 mm ID tube. CFX4-Meshbuild was used to define the geometry of the flow domain and generate the finite volume grid. Because of the geometrical symmetry only half of the pipe was modelled and this was subdivided into four blocks which were referred to as the *top block*, *side block*, *bottom block* and *middle block*, respectively as illustrated in Figure 1a. Figure 1b shows an example of the mesh generated at the XY plane. In the cross-sectional plane there are 600 sub-divisions (see Figure 1b) and in the longitudinal direction there are 200 sub-divisions making a total of 120,000 hexagonal finite volume cells. The cross-sectional sub-division has 10×10 cells in the top and bottom blocks and 20×10 cells in the side and middle blocks (see Figure 1b).

The pre-processing stage involves definition of the physical and fluid properties of the fluids and specification of the boundary conditions. The present model consisted of four types of patches on the boundary (i) *inlet* for input flow (ii) *mass flow boundary* for the output flow (iii) *symmetry* plane and (iv) *wall*.

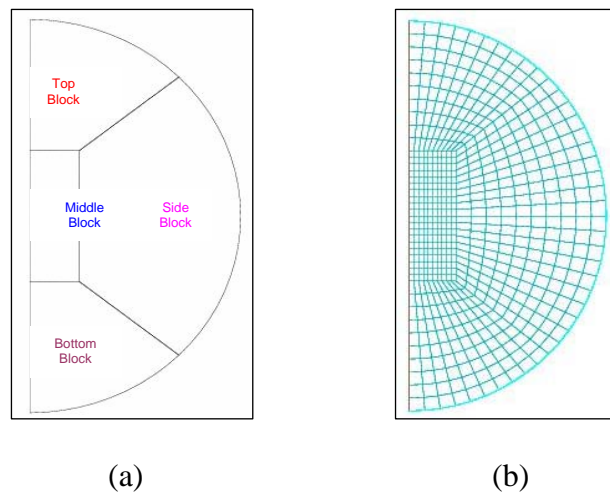


Figure 1. Topology and block distribution in half tube geometry.

VERTICAL PHASE DISTRIBUTION AND SAUTER MEAN DIAMETER

The main series of computations were carried out using CFX 4.4 with the MUSIG algorithm implemented and with the non-drag forces included. Essentially the main series of computations were carried out in three stages as follows:

- (1) *Uniform initial distributions of the dispersed phase and drop size.*

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. The results demonstrate the progressive separation of the phases along the channel which is also seen clearly by plotting the vertical distributions of chordal mean water fraction as shown in Figures 2. This separation is contrary to the experimental findings [12,18].

The MUSIG Population balance Model leads to predictions of the sizes and locations of the dispersed drops. Typical class and cumulative drop size distribution are given in Figures 3. As will be seen, there is a preponderance of smaller drops which will partly reflect the lower value of the coalescence factor chosen. Perhaps the most interesting output on drop size is that for the positional variation of Sauter mean diameter, starting with a uniform drop size, the drops near the wall become large and the drops in the centre became smaller. The area averaged Sauter mean diameter for a position 9.6 m from the inlet is shown in Figure 4 as a function of continuous phase velocity (mixture velocity divided by continuous phase fraction). The capability of the MUSIG algorithm of calculating the size distributions as a function of axial position is an interesting one but much more investigation and validation will be needed before reliable values can be obtained for liquid-liquid flows.

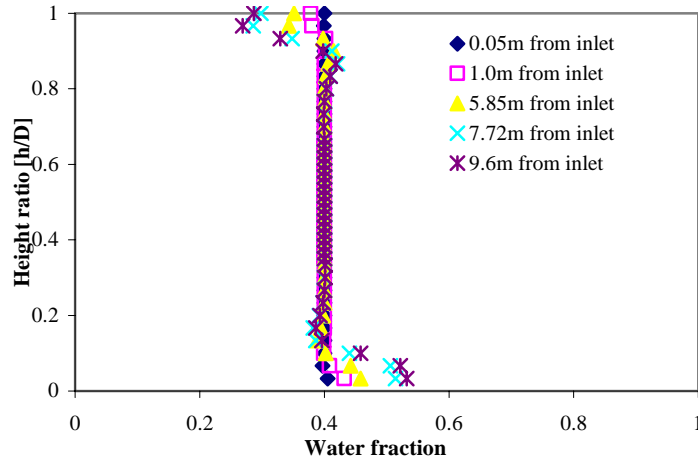


Figure 2. Vertical distribution of chordal mean water fraction as a function of position for 40% of input water fraction and a mixture velocity of 2.76 m/s.

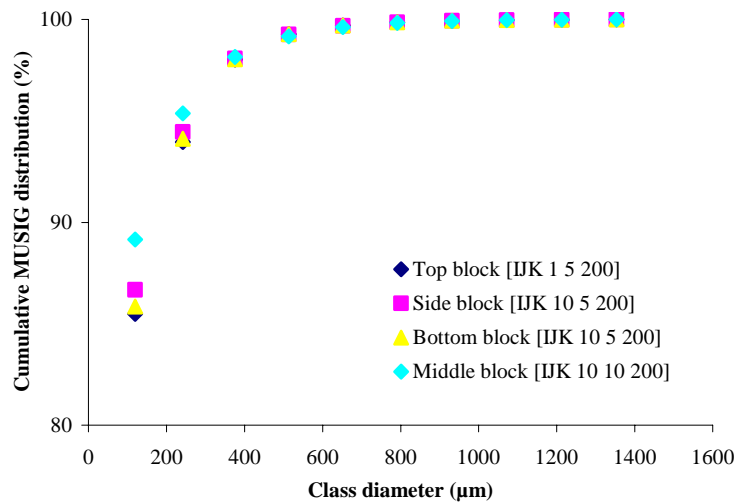


Figure 3. The cumulative MUSIG distribution against the class diameter at 7.72 m from inlet of the tube section 40% water fraction and 2.76 m/s mixture velocity.

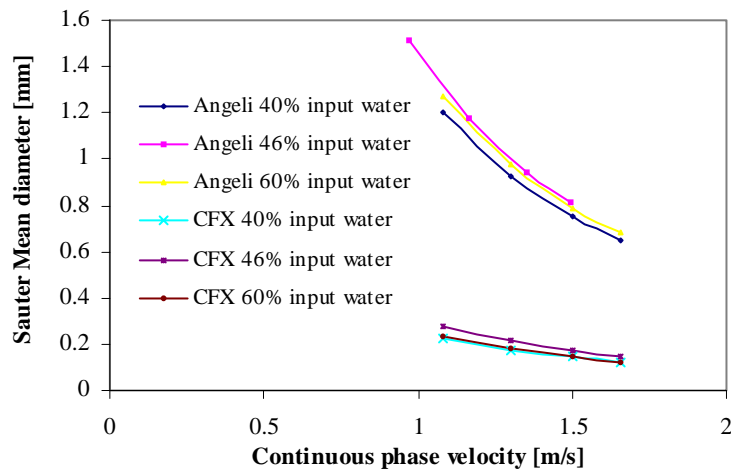


Figure 4. Comparison of calculated average Sauter mean diameter with values estimated with the Angeli's correlation.

(2) Non-uniform initial distribution of the dispersed phase.

This limited set of calculations was done to establish whether an initially partially separated flow would continue to separate or would tend to mix. The aim was to obtain some insight into what might happen in the experimental case where the flow was initially partially separated. The purpose of this calculation was to confirm that, starting with a non-uniform initial condition, the calculation would indicate continued separation. The calculations were done for a mixture velocity of 2.76 m/s and the initial condition was of a water fraction of 40% in the bottom block of the grid at the inlet [12,18]. The water fraction in the rest of the pipe was zero. Clearly separation is occurring with the water fraction at the bottom of the pipe rising. This is confirmed in the graphs of vertical chordal mean void fraction shown in Figure 5. The calculations were extended (though with some loss of accuracy due to the reduction in grid density) to a distance of 19.3 m from the inlet (i.e. much longer than actual test section) to a certain whether a steady state would be reached. The results indicate that such a steady state is calculated to be achieved for length of 11.7 m and longer. Again, these predictions are contrary to the experimental results which indicate a tendency towards mixing rather than separation at these velocities [12,18].

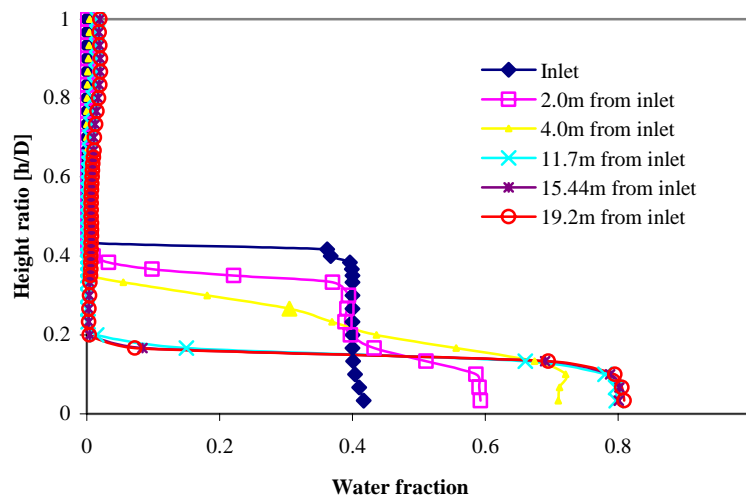


Figure 5. Vertical chordal mean water fraction distribution for the non-uniform input water fraction case at mixture velocity 2.76 m/s.

(3) *Initial conditions matching the experimental measurements at a distance of 1 m from the inlet.*

Though it was not possible to match the initial conditions precisely in the calculations, a compromise was adopted in which the experimental phase distribution at 1 m was used as a starting point for the CFD calculation. As a final demonstration of the differences between the experimental and predicted trends for phase distributions, the CFX calculation was started at 1.0 m with the inlet water fraction distribution matching that measured. The tomographic data were used to generate an input file for CFX. The experimental and CFX fitted distribution at 1.0 m matched well as illustrated in Figure 6-8 show the experimental and predicted distributions. The predictions show a tendency for phase separation whereas the experiments show a tendency towards complete mixing! This confirms the experience with the other calculations discussed above. A vast amount of data is generated by the calculation and only selected results are shown. The complete output files will be put as a record in the archives of the Department of Chemical Engineering and Chemical Technology at Imperial College.

The comparison of the CFX prediction with the experiments [12,18] reveals one overriding fact, namely that CFX predicts the evolution of separation whereas the experiments show the evolution of mixing. This is despite the fact that the adjustable parameters were set in CFX to somewhat favour mixing. The mixing processes assumed in CFX 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.

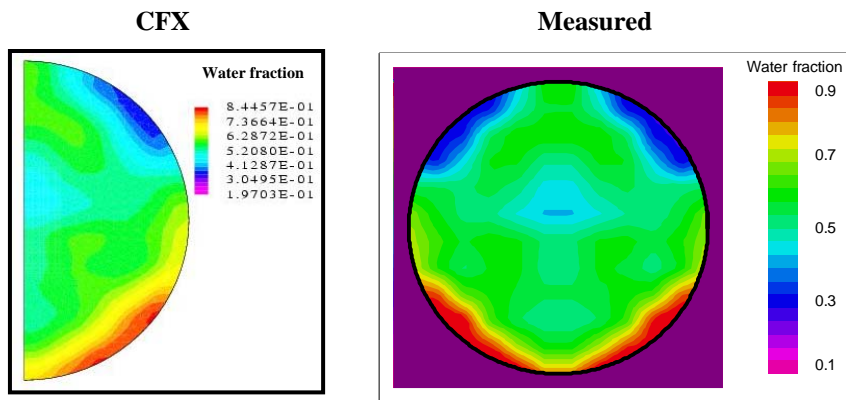


Figure 6. Comparison of vertical phase distribution between fitted CFX contours and tomography contours at a position 1.0 m from the inlet (input water fraction 40% mixture velocity 2.76 m/s).

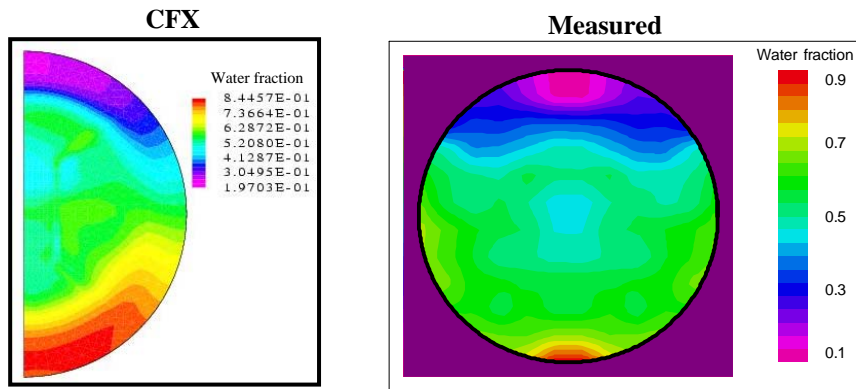


Figure 7. Comparison of measured cross-sectional phase distributions at a distance of 5.85 m from the inlet with that calculated using CFX using the measured water fraction data at 1.0 m as input (input water fraction 40%, mixture velocity 2.76 m/s).

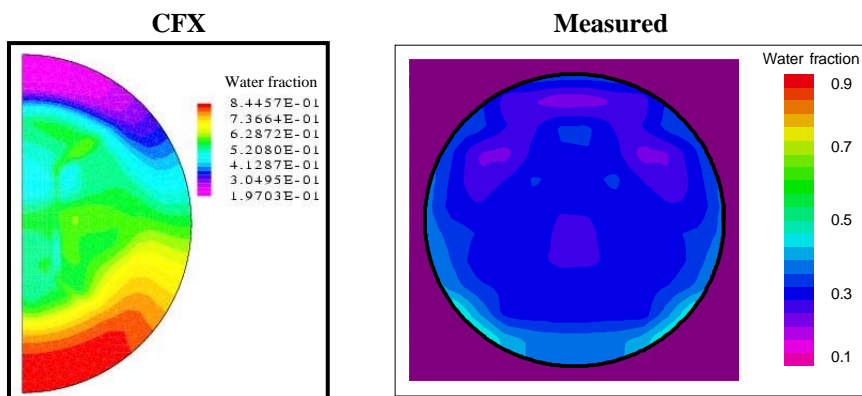


Figure 8. Comparison of measured cross-sectional phase distributions at a distance of 7.72 m from the inlet with that calculated using CFX using the measured water fraction data at 1.0 m as input (input water fraction 40%, mixture velocity 2.76 m/s).

CONCLUSION

A CFD model was set up for liquid-liquid flow in a horizontal cylindrical tube with 25.4 mm ID and 9.7 m in length similar to the experimental design. It was developed to compute the spatial phase distribution of the oil and water in dispersed flow. The geometrical symmetry allowed only half of the tube to be modeled and this was subdivided into four cross-sectional blocks (see Figure 1). The tube was divided into 200 axial cells and a grid with a total of 120,000 hexahedral cells resulted. The calculations were performed using CFX 4.4. A range of input conditions was investigated. The most important results from the computations are shown in Figure 6-8 which compares the predicted and actual evolution of the flow. Despite the selection of input parameters which would favour mixing, the computations indicate a tendency to separate whereas the experiments indicate a tendency to mix. This implies that another mechanism for mixing is present, perhaps Kelvin-Helmholtz instability.

ACKNOWLEDGEMENTS

The author wish to acknowledge with grateful thanks to Universiti Putra Malaysia, for the research grant and deepest gratitude to Professor Geoff F. Hewitt, Dr. Xu Yun Xu and Dr. Colin Hale for their consistent help, understanding, guidance, support, advice and encouragement.

REFERENCES

- [1] Angeli, P. (1996). Liquid-liquid in Horizontal Pipes. Ph.D Thesis, Imperial College. University of London. UK.
- [2] Angeli, P. and Hewitt, G.F. (2000). Drop Size Distributions in Horizontal Oil-Water Dispersed Flows. *Chem. Eng. Science*, **55**: 3133-3143.
- [3] Angeli, P. (2001). Droplet Size in Two-Phase Liquid Dispersed Pipeline Flows. *Chem. Eng. Technology*, **24**: 431-434.
- [4] Angeli, P. and Lovick, J. (2004). Droplet Size and Velocity Profiles in Liquid-liquid Horizontal. *Chem. Eng. Science*, **59**: 3105-3115.
- [5] CFX-4.3. (1999a). Examples. AEA Technology Engineering Software.
- [6] CFX-4.3. (1999b). Solver (Volume 1-3). AEA Technology Engineering Software.
- [7] CFX-4.3. (1999c). Pre-Processing (Volume 1-2). AEA Technology Engineering Software.
- [8] CFX-4.3. (1999d). Post-Processing. AEA Technology Engineering Software.
- [9] CFX-5. (2000). Post –Processing. AEA Technology Engineering Software.
- [10] Clift, R, Grace, J. R and Weber, M.E. (1978). *Bubbles, Drops and Particles*. Academic Press, Inc.
- [11] Hinze, J.O. (1955). Fundamentals of the Hydrodynamic Mechanism of Splitting in Dispersion Processes. *AIChE Journal*, **1**(3): 289-295.
- [12] Hussain, S.A. (2004). Experimental and Computational Studies of Liquid-Liquid Dispersed Flows. PhD Thesis. Imperial College. University of London. UK.
- [13] Lan Lui, Omar K. Matar and Geoffrey F. Hewitt. (2005). Computational Fluid Dynamic (CFD) Studies of Liquid-liquid Flows. Part 1: Calculations on Phase Distribution and Drop Size in Upwards Flows. Imperial College London. UK.
- [14] Lo, S. M.(1989). Mathematical Basis of a Multiphase Flow Model. Harwell Company.
- [15] Lo.S. M. (1996). Application of the MUSIG Model to Bubbly Flows. AEAT-1096, AEA Technology. June 1996.
- [16] Luo H. and Svendsen H.F. (1996). Theoretical Model for Drop and Bubble Breakup in Turbulent Dispersions. *AIChE Journal*, **42**, (5) :1225-1233.
- [17] Prince M.J. and Blanch H.W. (1990). Bubble Coalescence and Break-Up in Air-Sparged Bubble Columns. *AIChE Journal*, Oct 1990. Vol 36. No 10:1485-1499.
- [18] Siti Aslina, H. (2005). Multiple-Size-Group Model in Liquid-liquid Horizontal Flows. 2nd International Conference on Chemical and Bioprocess Engineering (SOMCHE 2005). Kota Kinabalu. Sabah.
- [19] Soleimani, A. (1999). Phase Distribution and Associated Phenomena in Oil-Water Flows in Horizontal Tubes. Ph.D Thesis. Imperial College. University of London. UK.