INTERNATIONAL JOURNAL OF RESEARCH IN AERONAUTICAL AND MECHANICAL ENGINEERING

ISSN (ONLINE): 2321-3051

Vol.3 Issue.5, May 2015. Pgs: 41-51

CFD ANALYSIS OF 2 FLUIDS FLOW OF AIR WATER MIXTURE IN A VERTICAL 2D PLANER T JUNCTION USING EULERIAN MULTIPHASE MODEL

Kriti Gupta[1], Mahendra Dawande[2] , Abhishek Tripathi[3]

[1]PG (Post Graduate) Student, Mechanical Engineering Department , Bansal Institute Of Research and Technology, Bhopal (M.P.) guptakriti.mech@gmail.com, +919425715462

[2] Assistent Professor, Mechanical Engineering Department, Bansal Institute Of Research and Technology, Bhopal(M.P.)

[3]Abhishek Tripathi, abhiyoho@gmail.com, +919630444300

Abstract

Multiphase flow is a common phenomenon in many industrial processes, amongst them the oil and gas industry. Due to the complexity of multiphase flow, development of reliable analysis tool is difficult. Computational fluid dynamics (CFD) has been an established tool for flow analysis in the field of single phase flow for more than 20 years but has only started to become established in the multiphase field as well. The purpose of this work is to analyze Eulerian- multiphase model available in the ANSYS software Fluent 14.5 and perform simulations using the Eulerian Phase model and standard k-Ɛ (epsilon) 2-eqn viscous turbulent method. Based on the simulations, it was evident that the Euler-Euler modeling approach was best suited for predicting the phase redistribution phenomenon in the T-junction. Multiphase flow is a generalization of the modeling used in two-phase flow to cases where the two phases are not chemically related (e.g. dusty gases) or where more than two phases are present (e.g. in modeling of propagating steam explosions).One of the typical and important objectives of this modeling and simulation analysis is maximize the contact between the different phases, typically different chemical compounds The standard k-epsilon viscous turbulent method was used in this work to analised the turbulency of multiphase flow in different sections. Multiphase flow is important in many industrial processes. The main industrial processes where multiphase flow is important are Riser reactors, Bubble Column reactors, Fluidized Bed reactors, Scrubbers, Dryers etc.

***Keywords**:* Multiphase, Computational fluid dynamics, T-junction, Phase redistribution, Euler-Euler.

1. Introduction

A phase can be defined as one of the states of matter such as gas or liquid, solid. Multiphase flow is the simultaneous flow of several phases, with two phase flow being the simplest case [1]. The analysis of fluid flow in T-junction is important for a number of engineering applications like piping system, air conditioning devices, pipe sections in industries etc. Due to lack of available theory and model for pressure drop calculation

Vol.3 Issue.5, May 2015. Pgs: 41-51

in multiphase flow [2], a computational fluid dynamics study has been performed for T-junction using Eulerian Multiphase Model and presented in this paper.

Several error sources exist for numerical simulations. Numerical approximation errors will always occur but another error source, which often is difficult to detect, is usage error. Unintended application of models, badly chosen parameters or wrongfully applied boundary conditions can lead to unphysical and inaccurate results [3]. Two-Phase Flow in a Mini-Size Impacting Tee Junction with a Rectangular Cross-Section[4], With the extended use of CFD simulations in engineering work it is of high importance to investigate the accuracy of commercial codes as well as understanding the choice of models. This is particularly important for multiphase flow where the complexity of both physical laws and numerical treatment makes the development of general models difficult [9].

1.1 Multiphase flow theory

Multiphase flow is flow with simultaneous presence of different phases, where phase refers to solid, liquid or vapor state of matter. There are four main categories of multiphase flows; gas-liquid, gas-solid, liquid-solid and three-phase flows. Further characterization is commonly done according to the visual appearance of the flow as separated, mixed or dispersed flow. These are called flow patterns or flow regimes and the categorization of a multiphase flow in a certain flow regime is comparable to the importance of knowing if a flow is laminar or turbulent in single-phase flow analysis [10]. A flow pattern describes the geometrical distribution of the phases and the flow pattern greatly affects phase distribution, velocity distribution and etc. for a certain flow situation. A number of flow regimes exist and the possible flow patterns differ depending on the geometry of the flow domain. For some simple shapes, for example horizontal and vertical pipes, the flow patterns that occur for different phase velocities etcetera have been summarized in a so called flow map. Figure 1.Visualizes the flow configuration for some possible flow regimes and Figure 2.Shows an example of a flow maps for horizontal pipe flow.

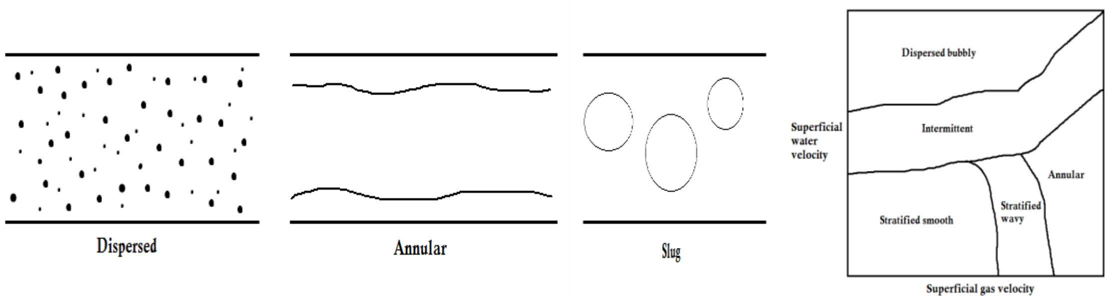


Figure. Example of typical flow patterns for flow in horizontal pipes & Example of flow map for two phase flow in horizontal pipes.

1.2 Euler-Euler approach

In Euler-Euler models all phases are treated as continuous. For that reason, these models are often also called multi-fluid models. Multi-fluid models are appropriate for separated flows where both phases can be described as a continuum. However, the Euler-Euler approach can also be used to model dispersed flows when the overall motion of particles is of interest rather than tracking individual particles. The dispersed phase equations are averaged in each computational cell to achieve mean fields. To be able to describe a dispersed phase as a continuum, the volume fraction should be high and hence this approach is suitable for dense flows. The phases are treated separately and one set of conservation equations are solved for each phase. Coupling between the phases is achieved through a shared pressure and interphase exchange coefficients. The interphase exchange coefficients need to be modeled. Just as in the Euler-Lagrange approach it is up to the modeler to decide which interphase phenomena to include.

A mixture model is a simplified version of an Euler-Euler model. As in the Euler- Euler models both phases are treated as interpenetrating continua but in the mixture model the transport equations are based on mixture properties, such as mixture velocity, mixture viscosity etc. The phases are allowed to move with different velocities by using the concept of slip velocity, which in turn includes further modeling.

2. COMPUTATIONAL FLUID DYNAMICS (CFD)

The flow equations are of coupled non-linear partial differential equations. These can be solved analytically only for very simple cases. For real life flows one must use numerical methods to transform the equations into algebraic approximations. In computational fluid dynamics numerical approximations of the solutions to the governing equations are obtained using computers.

2.1 The finite volume method

A method for discretising the transport equations commonly implemented in CFD codes is the finite volume method (FVM). In a FVM, the computational domain is divided in control volumes and conservation principles are applied to each control volume. This ensures conservation, both in each cell and globally in the domain, which is a great advantage of the FVM. Using FVM also allows for the use of unstructured grids which decreases the computational time.

2.2 Centre node based FVM

In a centre node based FVM, the computational domain is divided into a mesh where each element in the mesh makes up a control volume. The transport equations are integrated over each control volume and then discretised to obtain one set of algebraic equations for each control volume/cell. In the centre node based FVM approach, the value of each variable is stored in a node in the centre of the cell. However, the discretised equations also include values for the cell faces. Therefore, interpolation methods are used to obtain approximate values at these positions. The choice of interpolation method has a great impact on numerical stability, convergence rate and accuracy.

2.3 Fluent

The Fluent solver is based on the centre node FVM discretisation technique and offers both segregated and coupled solution methods. Three Euler-Euler multiphase models are available; the Eulerian model, the mixture model and the VOF model. In addition, one particle tracking model is available. Fluent offers three main

INTERNATIONAL JOURNAL OF RESEARCH IN AERONAUTICAL AND MECHANICAL ENGINEERING

ISSN (ONLINE): 2321-3051

Vol.3 Issue.5,

May 2015.

Pgs: 41-51

approaches to model dispersed phases with a two-fluid formulation. With the default settings it is assumed that the dispersed phase has a constant diameter or a diameter defined by a user-defined function. With this setting, phenomena such as coalescence and breakage are not considered.

3. METHODOLOGY

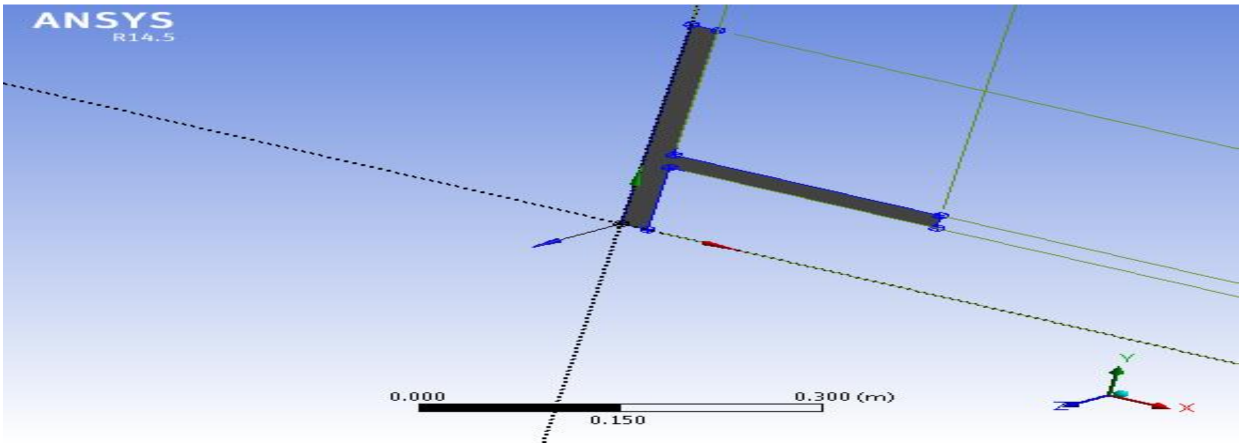
In this section the methodology is presented. Firstly, the geometry and mesh are discussed. This is followed by the choice of simulations settings and a description of the boundary conditions. Lastly, the convergence and evaluation criteria are presented.

3.1 The CFD Modeling

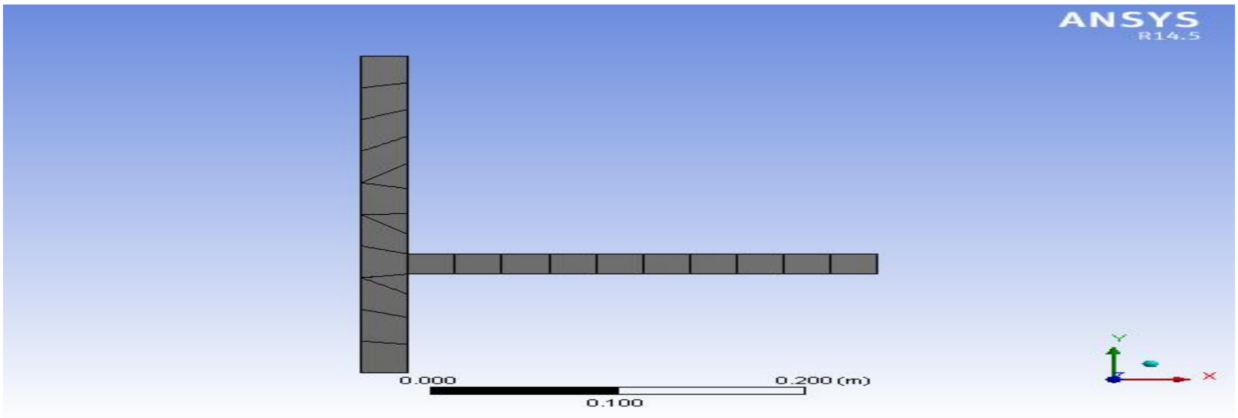
With the advancement and recent development of CFD codes, a full set of fluid dynamic and multiphase flow equations can be solved numerically. The current study used commercial CFD code, FLUENT [14.5], to solve the balance equation set via domain discretization, using control volume approach. These equations are solved by converting the complex partial differential equations into simple algebraic equations.
The κ-ε turbulence model with standard wall functions were used due to their proven accuracies in solving mixture problems. The gravitational acceleration of 9.81 m/s2 in upward flow direction was used.

3.2 Geometry Details

For present domain the inner diameter of the pipe was 25mm (0.025m) and pipe arms were made 250mm (0.25m) long.



3.3 Meshing



A block-structured meshing approach was used to create meshes with only hexahedral/quad cells. Refinements were made in the vicinity of the junction to resolve the rapid changes in the flow occurring there. The smallest elements were created at the wall to resolve boundary layers and the cell size increased by 10% in a radial direction. The distribution of cells at the inlet and around the junction can be seen in above Figure. In addition to the meshes for the three domains, refined meshes were also created to investigate the influence of mesh size. Statistics for all the meshes can be found in below table.

Table 3.1 Mesh statistics

Sizing	Relevance center	Use advanced size function	Initial size seed	Smoothing	Span angle center	Element midside nodes
	Coarse	On curvature	Active assembly	Medium	fine	Dropped
Inflation	Inflation option	Transition ratio	Maximum layers	Growth rate	Inflation algorithm	
	Smooth transition	0.272	2	1.2	Pre	
Statistics	Node = 59	Elements = 31				

4. SIMULATION SETTINGS

For the simulations ran in Fluent the pressure-based coupled solver was used with gravity enabled. The settings for the simulations in Fluent can be found in Table 4.1

Run	Settings
1.	Steady State Eulerian : air bubble diameter = 1mm air velocity = 1.6m/s , Volume fraction of air = 0.02
2.	Solver : pressure based
3.	Problem type : 2D-planer ,Gravity- On (9.81m/s)
4.	Models : Multiphase – Eulerian No. of Eulerian phase = 2 Viscous - k-2eqn turbulence model
5.	Materials : fluid – Air & water (from fluent database)
6.	Phases :Phase 1 - Primary phase – water Phase material – water-liquid Phase 2 - Secondary phase – air Phase material – air Cell zone condition : zone- surface body Phase – mixture Type – fluid

5. BOUNDARY CONDITION CONSIDERED

A summary of the boundary conditions can be found in Table 5.1

Inlet	1. Inlet Velocity – phase – mixture Turbulence intensity (%) = 10 Hydraulic diameter = 0.025m 2. Inlet velocity – phase – water, Velocity of water = 1.53m/s 3. Inlet velocity - phase – air Velocity of air = 1.6m/s Multiphase – volume fraction = 0.02

Outflow-1	Flow rate weighing = 0.38
Outflow-2	Flow rate weighing = 0.62
Operating condition	Pressure – default, Gravity – y= 9.81m/s
Solution methods	Pressure velocity coupling – Multiphase Coupled
Solution control	Explicit relexation factor - momentum = 0.5 Pressure = 0.5 Under relaxation factor – volume fraction = 0.4
Solution initialize	From inlet
Run calculation	No. of iteration = 1000

5.1 Convergence criteria

Convergence was judged based on three criteria. First of all, the normalized equation residuals for momentum, continuity, and turbulence and volume fraction equations were monitored and however, this criterion alone is not enough for judging the validity of the solution. For some cases the residual criterion might never be fulfilled even though the solution is valid and for other cases the solution can be incorrect even though the residuals are low. Therefore, mass conservation and outlet pressures were also monitored. The fractional difference in mass flow in and out the domain should be less than 0.01% and the mass flow through the open boundaries as well as the pressure at each outlet should remain constant for a number of iterations if the simulation would be said to be converged.

5.2 Evaluation criteria

The simulations were judged based on the following three conditions; accuracy, time requirement and numerical stability. The criterion for time requirement was straightforward. If a simulation needed little time to converge it was judged as good with respect to time and vice versa for long calculation times. To judge the accuracy of the results, the prediction of known flow phenomena and/or known values/features was evaluated. For the third criterion, the numerical stability, evaluation was based on how difficult it was to obtain a converged solution. If it was needed to reduce under relaxation factors and/or use other methods for stabilising the solution to achieve convergence the simulation was judged as less good with respect to stability.

6. RESULTS & DISCUSSION

In this section, the results from the simulations and from the boundary conditions are presented.

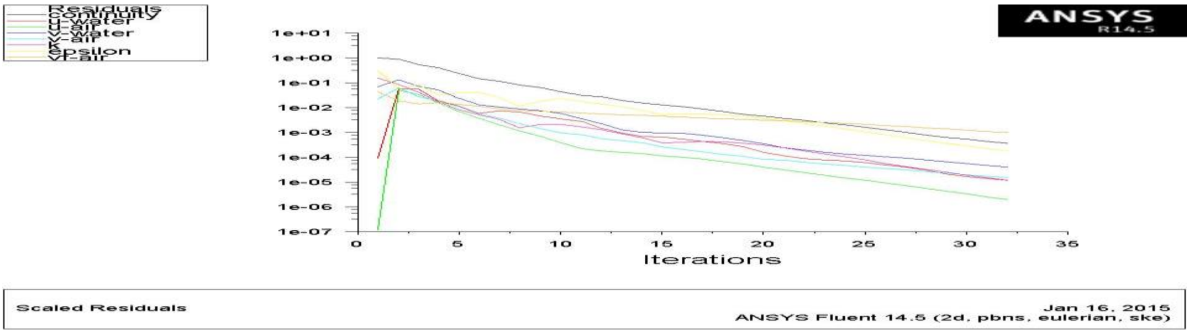


Figure 6.1 Converged multiphase model

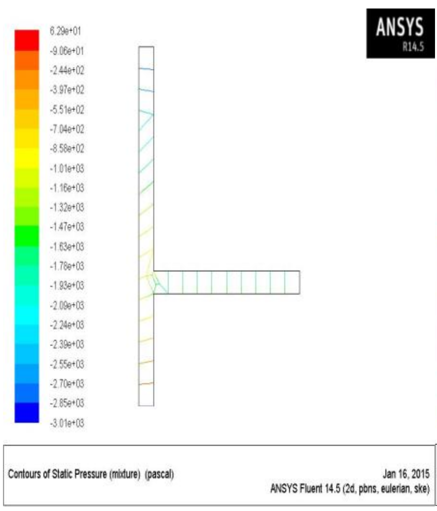


Fig 6.2 contour plot of pressure on mixture

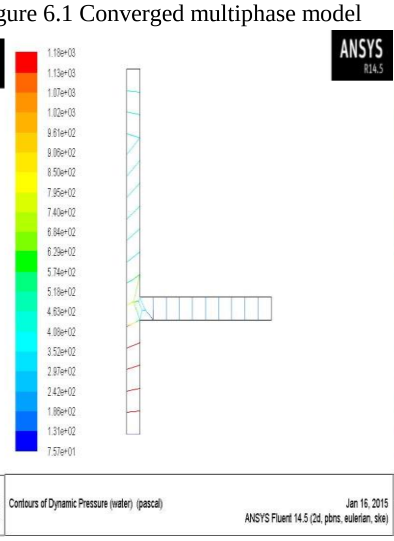


Fig 6.3 contour plot of pressure on water

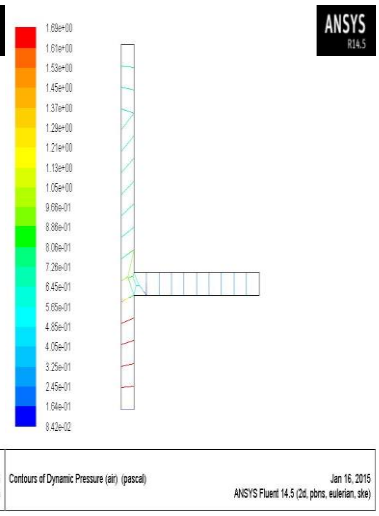


Figure 6.4 contour plot of pressure on air

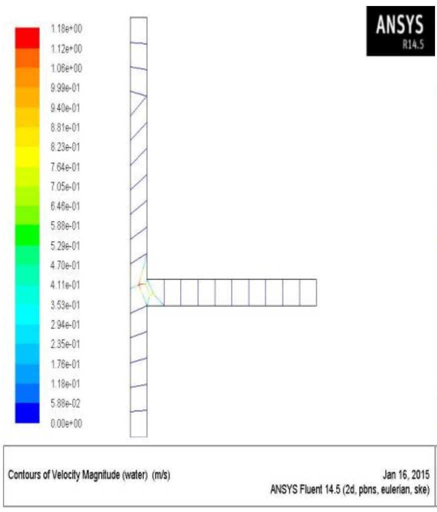


Fig 6.5 contour plot of velocity of Water

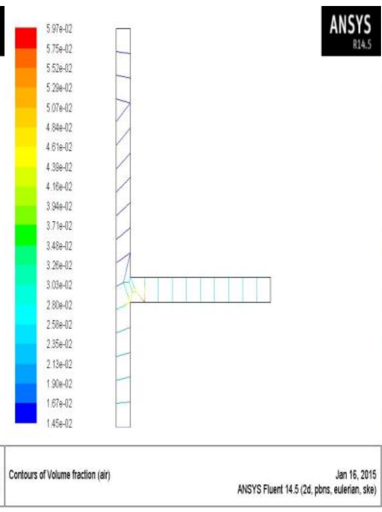


Fig 6.6 contour plot of volume fraction of air

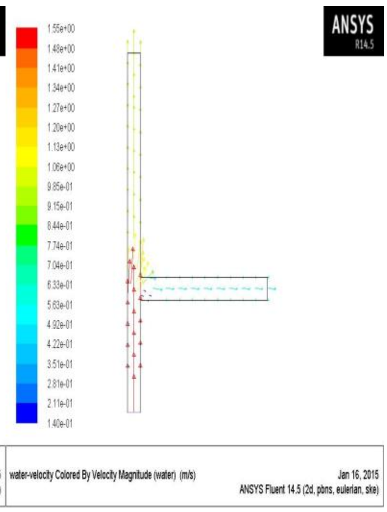


Fig 6.7 water velocity vector

Vol.3 Issue.5,

May 2015.

Pgs: 41-51

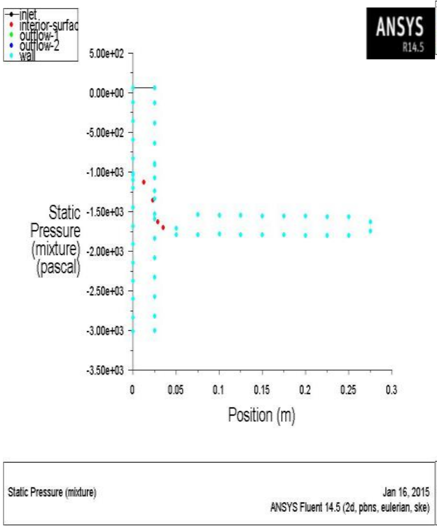


Fig 6.8 XY plot of pressure variation along position X (m)

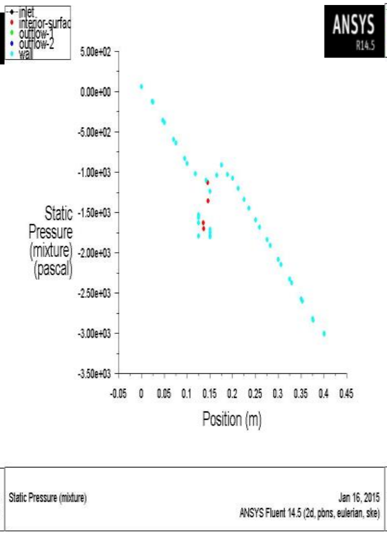


Fig 6.9 XY plot of pressure variation along position Y (m)

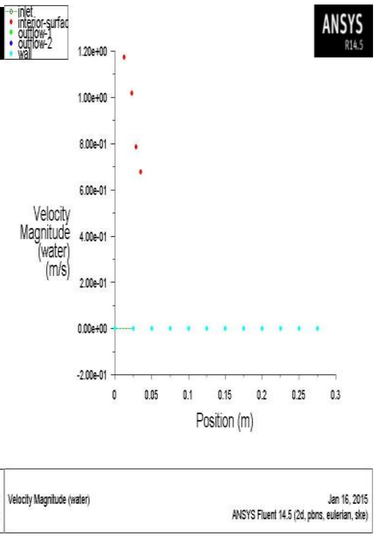


Fig 6.10 XY plot of velocity along position X (m)

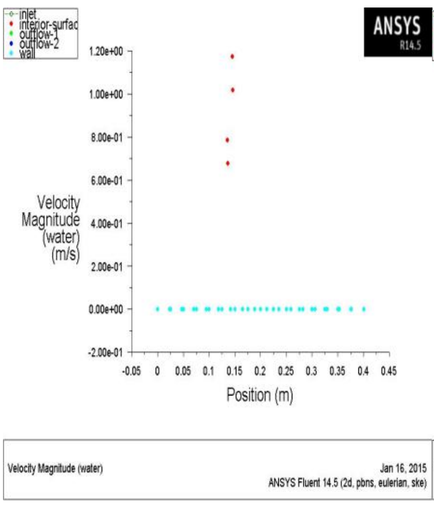


Fig 6.11 XY plot of velocity along position Y(m)

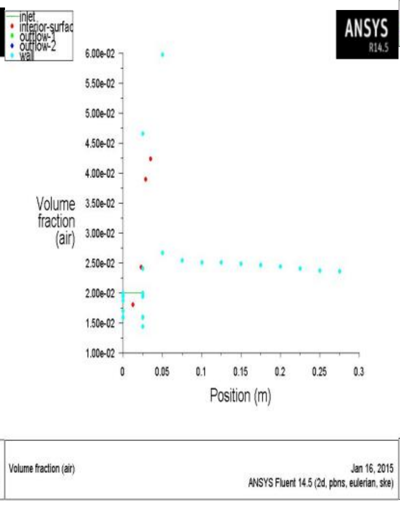


Fig 6.12 XY plot of volume fraction of air along position X(m)

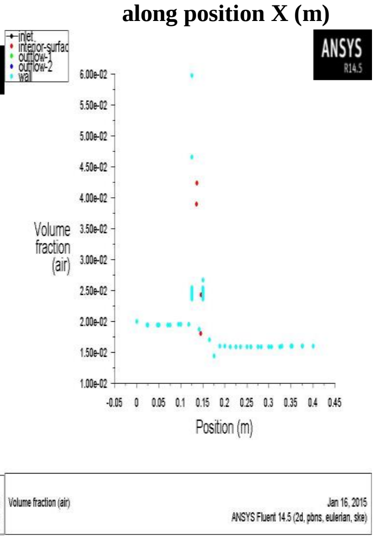


Fig 6.13 XY plot of volume fraction of air along position Y(m)

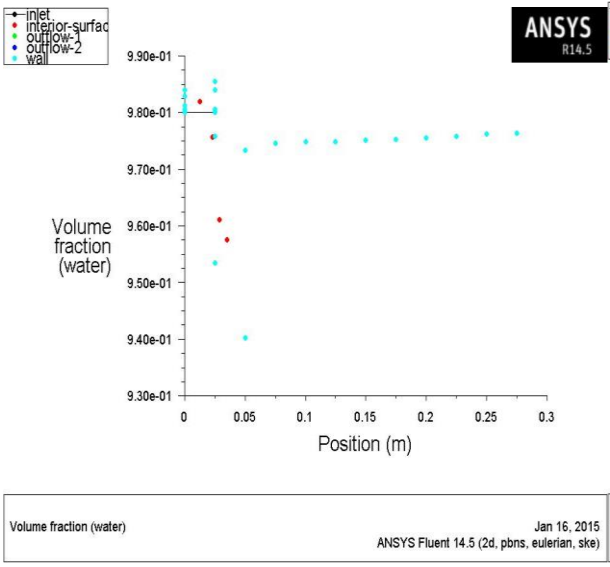


Fig 6.14 XY plot of volume fraction of water along position X(m)

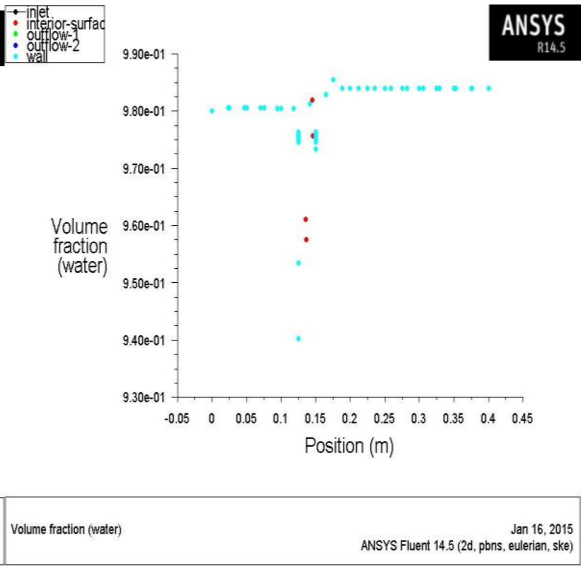


Fig 6.15 XY plot of volume fraction of water along position Y(m)

7. CONCLUSIONS

The purpose of this analysis (i.e. multiphase models offered in the ANSYS CFD software Fluent) was to investigate the effect of changing simulations parameters through numerical simulations. From the numerical simulations it is evident that the Euler-Euler approach is best suited for the T-junction application. Using an Euler-Euler model the flow redistribution phenomenon was captured in all cases, with the exception of an erroneous prediction of water separation in the run arm. In conclusion, the choice of simulation parameters is not crucial if only the flow phenomenon is of interest, however, for a more detailed analysis the effect of particle diameter and polydispersity was found the be strong. Generally, it can be concluded that there is a very complex interaction between the phases as well as the simulation parameters when performing numerical multiphase simulations. The importance of specific parameters changes depending on the application studied and it is therefore hard to give any general recommendations on the choice of parameters/settings.

References

1. Arlrachakaran, S.J., Brill, J.P. (1992): State of Art in Multiphase Flow, *Journal of Petroleum Technology*, Vol. 44, No. 5, May 1992, pp. 538-541.
2. Crowe, C.T., Schwarzkopf, J.D., Sommerfield and M., Tsuji, Y. (2012): Multiphase Flows with Droplet and Particles 2nd edition. CRC Press, Taylor & Francis Group, Boca Raton, Florida
3. N. M. Crawford, G. Cunningham and P. L. Spedding, “Prediction of Pressure Drop for Turbulent Fluid Flow in 90° Bends,” Proceedings of the Institution of Mechanical Engineers, Vol. 217, No. 3, 2003, pp. 153-155.

4. Elazhary, A.M. (2012): Two-Phase Flow in a Mini-Size Impacting Tee Junction with a Rectangular Cross-Section, Ph.D. Thesis. Department of Mechanical and Manufacturing Engineering, University of Manitoba, Winnipeg, Manitoba
5. Issa, R.I. and Oliveira, P.J. (1993): Numerical Prediction of Phase Separation in Two- Phase Flow through Junctions. Computers Fluid, Vol. 23, No. 2, February 1993, pp. 347-372.
6. Liu, Y. and Li, W.Z. (2011): Numerical Simulation on Two-Phase Bubbly Flow Split in a Branching T-junction, International Journal of Air-Conditioning and Refrigeration, Vol. 19, No.4, December 2011, pp. 253-262.
7. R. W. Lockhart and R. C. Martinelli, "Proposed Correlation of Data for Isothermal Two Phase Two-Component Flow in Pipes," Chemical Engineering Progress, Vol. 45, No. 1, 1949, pp. 39-48.
8. Walters, L.C., Soliman, H.M. and Sims, G.E. (1998): Two-Phase Pressure Drop and Phase Distribution at Reduced Tee Junctions, International Journal of Multiphase Flow, Vol. 24, No. 5, August 1998, pp. 775-792.
9. Van Wachem, B.G.M. and Almstedt, A.E. (2003): Methods for Multiphase Computational Fluid Dynamics, Chemical Engineering Journal, Vol. 96, No. 1,December 2003, pp. 81-98.
10. Thome, J.R. (2004): Engineering Data Book III, Wolverine Tube Inc., Decatur,Alabama, USA