IJRAME

INTERNATIONAL JOURNAL OF RESEARCH IN AERONAUTICAL AND MECHANICAL ENGINEERING
WWW.IJRAME.COM
ISSN (ONLINE): 2321-3051

Vol.5 Issue.6,
June 2017
Pg: -77-86

CFD INVESTIGATION OF MULTIPHASE FLOW BY EULERIAN MODEL FOR PLANER SECTION UNDER IMPLICIT CONDITIONS

Atul Singh[1], **S.K. Bharti**[2], **N.V. Saxena**[3]

[1]*Post Graduate (PG) student, Mechanical Engineering Department, Millenium institute of technology, Bhopal (M.P.), India, Telephone: 09200307160, email-singhatulmech@gmail.com*

[2]*Assistant Professor, Mechanical Engineering Department, Millenium institute of technology, Bhopal (M.P.)*

[3]*Assistant Professor, Mechanical Engineering Department, Millenium institute of technology, Bhopal (M.P.)*

Abstract

The aim of this present study was to investigate the effects of multiphase fluid flows in planer section by adopting eulerian – eulerian multiphase flow model. The present work is done on computational fluid dynamics as Fluent was the solver for whole work. Ideal situation for planer section is to be carried out for modeling and simulation and investigation is done on effects of volume fraction on section for defined number of iterations. 2D model of section is developed in fluent and then it is imported into meshing and solver for completing the simulation process. The total number of cells/elements was 398010 with orthogonal grid. Eulerian Multiphase Model was used to predict the vector and the scalar flow profile of liquid-liquid interaction in the planer section. A multiphase flow model of water and air, implicit unsteady profile and segregated flow model together with the laminar model are used in the simulation. Results were obtained in the form contour of volume fraction and velocity vector profile. These results are used to explain the behavior, how flow of fluid spreading water in pipe over time.

Keywords: *Multiphase flow, Eulerian model, Planer T section, Implicit conditions.*

1. Introduction

The term multiphase flow is used to refer to any fluid flow consisting of more than one phase or component. The flows considered some level of phase or component separation at a scale well above the molecular level. This still leaves an enormous spectrum of different multiphase flows. One could classify them according to the state of the different phases or components and therefore refer to gas/solids flows, or liquid/solids flows or gas/particle flows or bubbly flows and so on. Some treatises are defined in terms of a specific type of fluid flow and deal with low Reynolds number suspension flows, dusty gas dynamics and so on. Others focus attention on a specific application such as slurry flows, cavitating flows, aerosols, debris flows, and fluidized beds and so on. Multiphase flows are also a ubiquitous feature of our environment whether one considers rain, snow, fog, avalanches, mud slides, sediment transport, debris flows, and countless other natural phenomena to say nothing of what happens beyond our planet. Very critical biological and medical flows are also multiphase, from blood flow to semen to the bends to lithotripsy to laser surgery cavitation and so on. No single list can adequately illustrate the diversity and ubiquity. Consequently any attempt at a comprehensive treatment of multiphase flows is flawed unless it focuses on common phenomenological themes and avoids the temptation to digress into lists of observations. Two general topologies of multiphase flow can be usefully identified at the outset, namely disperse flows and separated flows. By disperse flows we mean those consisting of finite particles, drops or bubbles (the disperse phase) distributed in a connected volume of the continuous phase. On the other hand separated flows consist of two or more continuous streams of different fluids separated by interfaces.

1.1 Multiphase flow models

There are three ways in which such models are explored: (1) experimentally, through laboratory-sized models equipped with appropriate instrumentation, (2) theoretically, using mathematical equations and models for the flow, and (3) computationally, using the power and size of modern computers to address the complexity of the flow. In disperse flows two types of models are prevalent, trajectory models and two-fluid models. In trajectory models, the motion of the disperse phase is assessed by following either the motion of the actual particles or the motion of larger, representative particles. The details of the flow around each of the particles are subsumed into assumed drag, lift and moment forces acting on and altering the trajectory of those particles. Euler-Euler models, Lagrangian Model, volume of fluid model are the models which are used in multiphase flow analysis. These are the most commonly and important models for multiphase fluid flow.

1.2 Interaction with turbulence

Turbulent flows of a single Newtonian fluid, even those of quite simple external geometry such as a fully-developed pipe flow, are very complex and their solution at high Reynolds numbers requires the use of empirical models to represent the unsteady motions. It can also result in particle agglomeration or in particle fission, especially if the particles are bubbles or droplets. It is self-evident that the addition of particles to such a flow will result in complex unsteady motions of the particles that may result in non-uniform spatial distribution of the particles and, perhaps, particle segregation. It can also result in particle agglomeration or in particle fission, especially if the particles are bubbles or droplets.

2. Bubble or Droplet Translation

The fluid stresses due to translation may deform the bubbles, drops or deformable solid particles that make up the disperse phase; we should consider not only the parameters governing the deformation but also the consequences in terms of the translation velocity and the shape. We concentrate here on bubbles and drop in which surface tension, S, acts as the force restraining deformation. However, we will realize that there would exist a similar analysis for deformable elastic particles. Furthermore, the discussion will be limited to the case of steady translation, caused by gravity. Clearly the results could be extended to cover translation due to fluid acceleration by using an effective value of gravitation.

2.1 Multiphase flow patterns

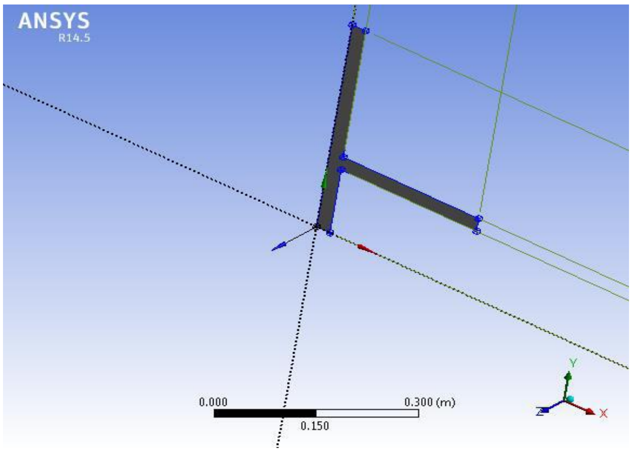
A particular type of geometric distribution of the components is called a flow pattern or flow regime and many of the names given to these flow patterns (such as annular flow or bubbly flow) are now quite standard. Usually the flow patterns are recognized by visual inspection, though other means such as analysis of the spectral content of the unsteady pressures or the fluctuations in the volume fraction have been devised for those circumstances in which visual information is difficult to obtain. For some of the simpler flows, such as those in vertical or horizontal pipes, a substantial number of investigations have been conducted to determine the dependence of the flow pattern on component volume fluxes on volume fraction and on the fluid properties such as density, viscosity, and surface tension. The results are often displayed in the form of a flow regime map that identifies the flow patterns occurring in various parts of a parameter space defined by the component flow rates. The flow rates used may be the volume fluxes, mass fluxes, momentum fluxes, or other similar quantities depending on the author. Perhaps the most widely used of these flow pattern maps is that for horizontal gas/liquid flow.

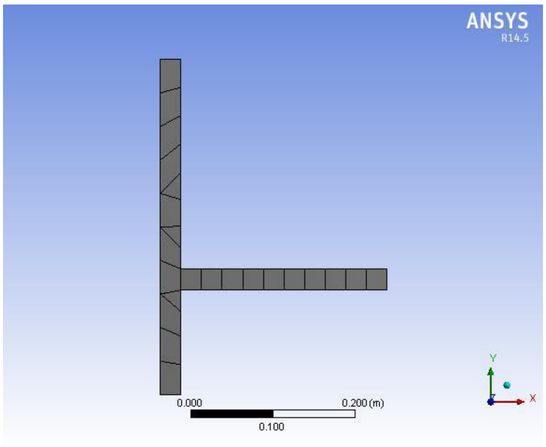
3. Methodology

2D modeling of T planer section and arrangements is done on Ansys software and imported into CFD FLUENT 14.5. For modeling 3GB ram, Pentium dual core processor system configurations were used and for thermal Effects and analysis of different parameters 3GB ram, Intel i3 processor system configurations were used.

3.1 Meshing details

Relevance center	Fine
Smoothing	Medium
Transition	Slow
Transition ratio	0.272
Maximum layers	5
Growth rate	1.2
Nodes	11240
Elements	5216
Minimum edge length	2.e-003m
Curvature normal angle	18.0^0





3.2 Solution setup

(i) General -

Solver – pressure based
Time – steady

(ii) Models -

Energy Equation – on
Standard K- epsilon model
C1- epsilon = 1.44
C2- epsilon = 1.92
Energy Prandtl no. = 0.85
Wall prandtl no. = 0.85

(iii) Materials -

Fluid – air & water
Solid – aluminium

(iv) Boundary Conditions - Inlet Velocity – phase – mixture

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

(v) Solution initialize From inlet

(vi) Run calculation No. of iteration = 1000

4. Results

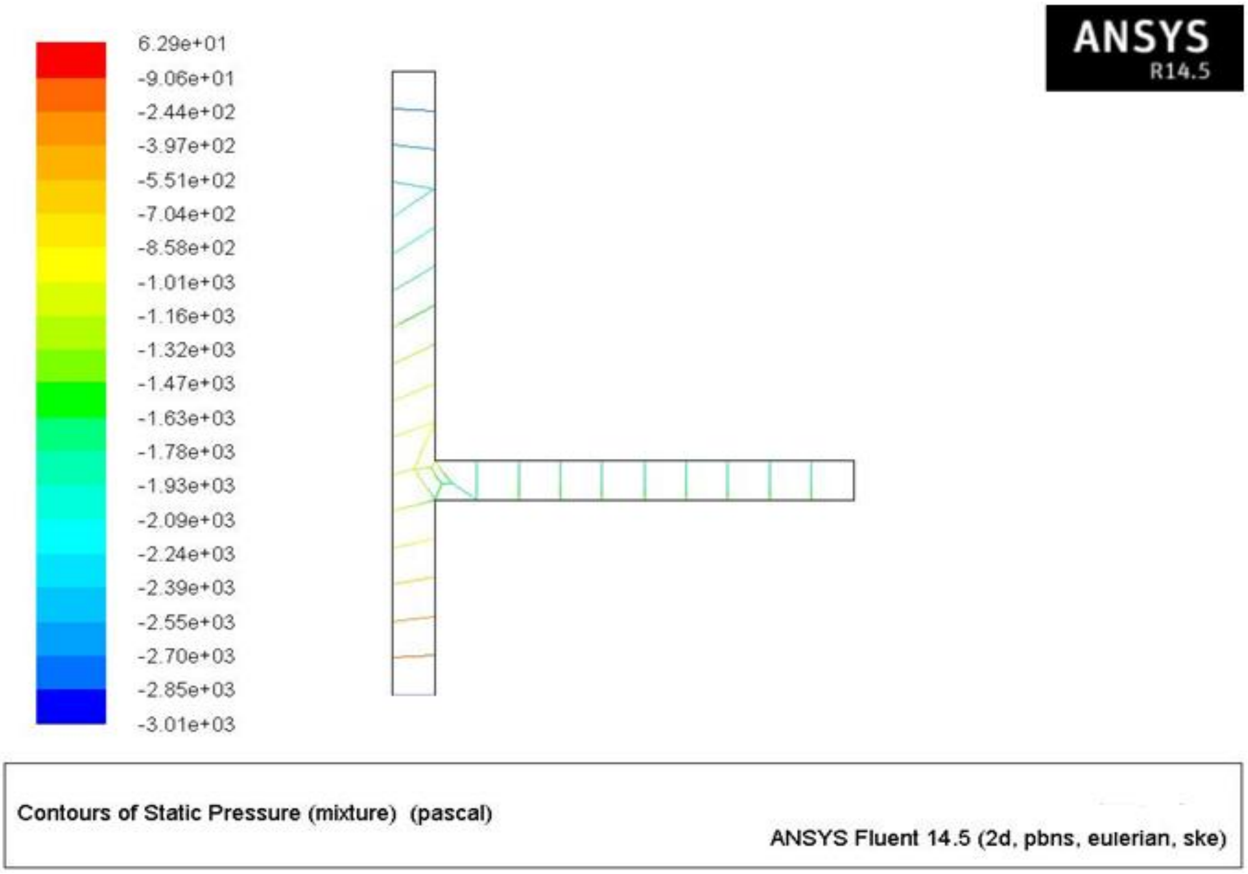


Figure 4.1 Contour plot of pressure on mixture

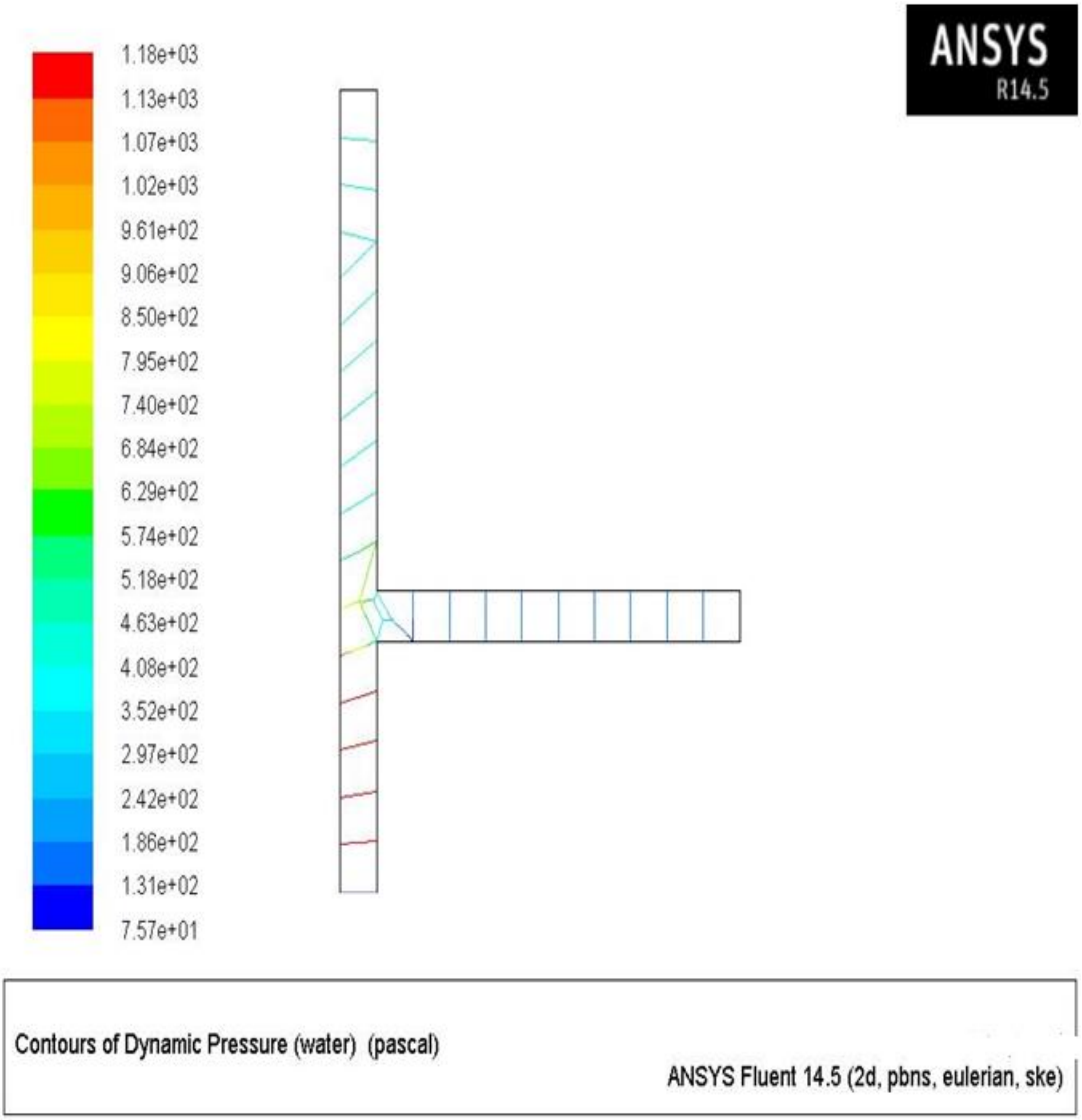


Figure 4.2 Contour plot of pressure on water

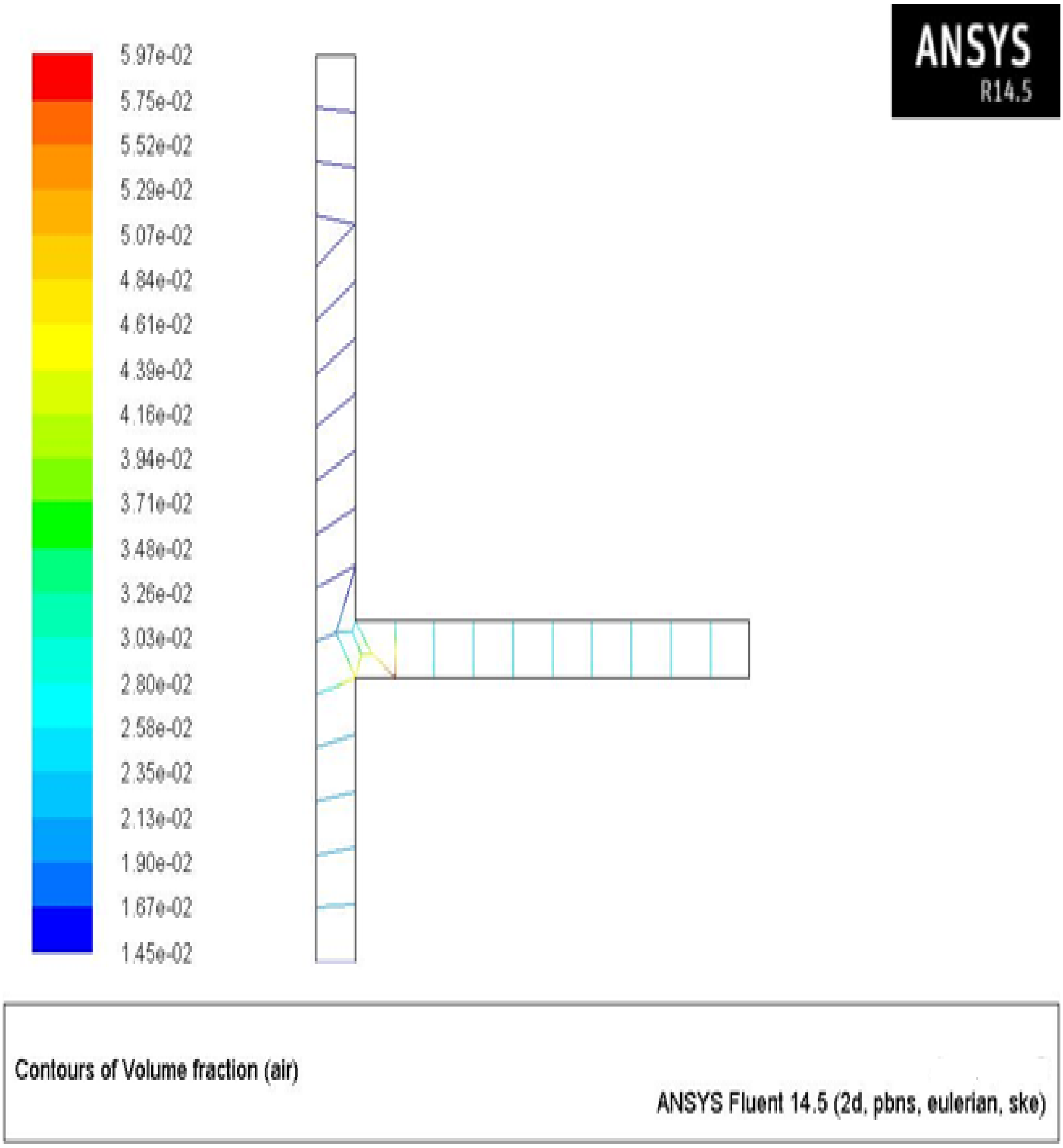


Figure 4.3 Contour plot of volume fraction of air

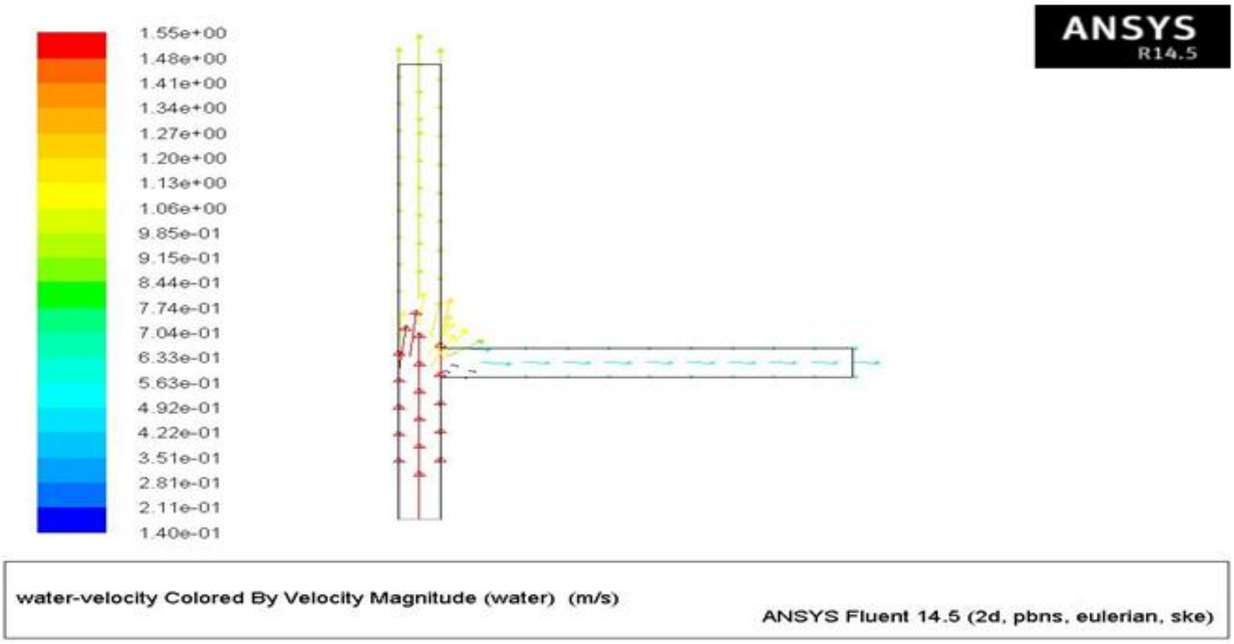


Figure 4.4 Water velocity vector

5. Conclusion

Outcome of the investigation and analysis which is done in ANSYS FLUENT 14.5 solver are as Follows:

- Static Pressure increases when mixture of fluid flows towards Outlet.
- 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 pipe.
- There is a very complex interaction between the phases as well as the simulation parameters when performing numerical multiphase simulations.
- 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 Dispersed phase was found the be strong.
- Velocity of water gives immense effects on mid section of pipe in by using Eulerian approach in multiphase model. In VOF (volume of fluid) model water flows like a constant velocity in defined inlet velocity conditions in solver setup.

6. Future Scope

As concluded, multiphase flow simulations involve a large number of parameters and models and due to the limited timeframe many of these parameters have not been investigated in this study. To propose models/settings resulting in better agreement with the experimental data, especially regarding local profiles, a further study could be made on the polydispersed modeling as well as on the effect of adding other forces of interest.

References

[1] Athulya A.Sa, Miji Cherian Rb , CFD Modelling of Multiphase Flow through T International Conference on Emerging Trends in Engineering, Science and Technology, Procedia Technology 24 (2016) 325 – 331

[2] X.J. Shi, P. Zhang, Two-phase flow and heat transfer characteristics of tetra-n-butyl ammonium bromide clathrate hydrate slurry in horizontal 90 elbow pipe and U-pipe,International Journal of Heat and Mass Transfer 97 (2016) 364–378

[3] Stephen Ambrose, Ian S. Lowndes, David M. Hargreaves,Barry Azzopardi,Numerical modelling of the rise of Taylor bubbles through a change in pipe diameter, Computers and Fluids 148 (2015) 10–25

[4] Cai L., Pan Z., Xiaoping C., and Changsui Z., Flow characteristics and stability of dense-phase pneumatic conveying of pulverized coal under high pressure., Experimental Thermal and Fluid Science 41 (2014)149-157

[5] Jing R., Ren F., and Wang X, The resistance properties of gas-solid flow for horizontal branch pipe. Advanced Materials Research (2012)361-363, 887–890.

[6] Wang Y., Williams K.C., Jones M.G., and Chen B, CFD simulations of gas-solid flow in dense phase bypass pneumatic conveying using the Euler-Euler model. International Communications in Heat and Mass Transfer . Applied Mechanics and Materials(2010) 26-28, 1190–1194.

[7] Gu Z. and Guo L, Simulation of horizontal slug-flow pneumatic conveying with kinetic theory. Frontiers of Energy and Power Engineering in China ,(2007) 336–340.

[8] Eskin D., Leonenko Y., and Vinogradov O, An engineering model of dilute polydisperse pneumatic conveying. Chemical Engineering and Processing 46(2007), 247–256.

[9] Heinl E. and Bohnet M ,Calculation of particle-wall adhesion in horizontal gas-solids flow using CFD Powder Technology 159(2005), 95–104.

[10] Zhu K., Wong C.K., Rao S.M., and Wang C.H, Pneumatic conveying of granular solids in horizontal and inclined pipes. AIChE Journal 50(2004), 1729–1745.

[11] C. Thianpong, K. Yongsiri, K. Nanan& S. Eiamsa-ard, Thermal performance evaluation of heat exchangers fitted with twisted-ring turbulators, International Communications in Heat and Mass Transfer 39 (2012) 861–868.

[12] P. Murugesan, K. Mayilsamy , S. Suresh & P.S.S. Srinivasan, Heat transfer and pressure drop characteristics in a circular tube fitted with and without V-cut twisted tape insert, International Communications in Heat and Mass Transfer 38 (2011) 329–334.

[13] S. Pethkool, S. Eiamsa-ard , S. Kwankaomeng& P. Promvonge, Turbulent heat transfer enhancement in a heat exchanger using helically corrugated tube, International Communications in Heat and Mass Transfer 38 (2011) 340–347.

[14] S.Eiamsa-ard, K.Wongcharee, P.Eiamsa-ard, C.Thianpong, Heat transfer enhancement in a tube using delta-winglet twisted tape inserts, ; Applied Thermal Engineering 30 (2010) 310-318.

A Brief Author Biography

1st Author

Mr. Atul Singh, PG Student of Mechanical Engineering Department, Millenium institute of technology, Bhopal under RGPV University.