CFD Simulation of Two Phase Flow and Heat Transfer in Helical Pipe with PBM model.

Rajkumar Rathod[1*],Dr. Omprakash Hebbal[2]
1* PG Student ,Thermal Power Engineering, PDA College of Engineering, Gulbarga585102, Karnataka (INDIA)
Professor[2] ,Department of Mechanical Engineering, PDA College of Engineering,Gulbarga585102, Karnataka (INDIA)

ABSTRACT

Two-phase flow and heat transfer in helical pipes were simulated using computational fluid dynamics and population balance model. Heat transfer to flow of viscosity grade oil 32 (VG 32)-air mixture through a helical pipe, at constant pipe wall temperature, was considered in this study. The bubble size distribution and coalescence and breakage of bubble groups handled through the population balance model (PBM). Important flow quantities, for air-VG 32 system, such as local void fraction, VG 32 oil velocity, and pressure drop and temperature distribution were calculated. Available correlations, and show the weakness of such correlations in the case of flow accompanied with heat transfer. It was found that centrifugal forces play an important role in the phase, velocity, pressure, and temperature distribution in helical pipe. CFD-PBM model has more ability to capture the accurate flow features.

Keywords: *Helical Pipes; Air–VG 32 oil, Two Phase Flow; Population Balance Modelling, CFD.*

1. Introduction

Helically coiled pipes are used widely in heat exchangers, steam generators, food and chemical plants, etc. Helical pipes heat exchangers have many benefits than straight pipe heat exchangers, because of their real-world importance of ease of manufacture and arrangement, higher efficiency in heat transfer and compactness in structure.

In many cases, the fluid flowing through the helical coils is in the form of two phase flow. Compared to single-phase flow, two-phase flow characteristics and frictional pressure drop are more complex and important for engineering practice. In dispersed gas-liquid flows, the bubble size distribution plays an important role in the phase structure and interphase forces, which, in turn determine the multiphase hydrodynamic behaviours, including the spatial profiles of the gas fraction, gas and liquid velocities, and mixing and heat and mass-transfer behaviours. These influences must be taken into account for obtaining good predictions in wide operating conditions when using computational fluid dynamics (CFD) simulation. The population balance model (PBM) is an effective technique to simulate the bubble size distribution .The pipe curvature causes centrifugal forces to act on the flowing fluid, resulting in a secondary flow pattern perpendicular to the main axial flow. This secondary flow pattern generally consists of two vortices, which move fluid from the inner wall of the tube across the centre of the tube to the outer wall. Upon reaching the outer wall it travels back to the inner wall following the wall

Many studies have been reported in practical use and many theoretical and experimental results have been reported . Compared to the several investigations of the single-phase heat transfer, only a few works on the two-phase heat transfer characteristics in helical pipes have been reported, a review of them reported in references. However, a few works were reported on the CFD modelling of the flow and heat transfer in helical pipes. Numerical simulations of incompressible turbulent flow in helical and curved pipes are presented by Friedrich et al. [4, 5]. They considered only a small portion of pipe, 7.5 diameters long, their works would be inadequate to resolve travelling waves, since was modelled with periodic boundary conditions and in fact travelling waves are not mentioned in these works.

D. G. Prabhanjan, G. S. V. Ragbavan and T. J. Kennic [1] studied to determine the relative advantage of using a helically coiled heat exchanger versus a straight tube heat exchanger for heating liquids. It's found that that the heat transfer coefficient was affected by the geometry of the heat exchanger and the temperature of the water bath surrounding the heat exchanger.

Timothy J. Rennie, Vijaya G.S. Raghavan [2] conducted numerical analysis on double-pipe helical heat exchanger modeled for laminar fluid flow and heat transfer characteristics under different tube sizes and fluid flow rates. The overall heat transfer coefficients were calculated for both parallel flow and counter flow. Validation of results was done by comparing with literature and the annulus Nusselt number was correlated with a modified Dean number which showed a strong linear relationship.

J.S Jayakumara et al. [3] conducted CFD analysis of single phased flow inside helically coiled tubes. Detailed analysis of fluid motion revealed that the fluid particles undergo oscillatory motion inside the pipe and this causes fluctuation in heat transfer rates. The correlation developed for estimation of average Nusselt Number and is found to be within reasonable limits

.Zheng et al. [4] applied a control-volume finite difference method having second-order accuracy to solve the three-dimensional governing equations. The laminar forced convection and thermal radiation in a participating medium inside a helical pipe were analyzed. By the numerical including and not including thermal radiation, the effects of Thermal radiation on the convective heat transfer was investigated. They found that the thermal radiation could enhance the total heat transfer rate.

Dr. K. E. Reby Roy [5] helical ducts are used in a variety of applications including food processing, thermal processing plants and refrigeration. They are advantageous due to their high heat transfer coefficient and compactness compared to straight tubes. The curvature of the coil governs the centrifugal force resulting in development of secondary flow i.e. the fluid stream in the outer side of the pipe moves faster than the fluid streams in the inner side of the pipe. In the present study, computational fluid dynamics (CFD) simulations using Fluent 6.3.26 are carried out for helical rectangular ducts wound over a cylindrical passage. The cylindrical passage is oriented horizontally and acts as a counter flow heat exchanger. The analysis is done by changing the flow rates of four different fluids like Ethylene Glycol, Kerosene, Nano Fluid and water. The fluid flow and heat transfer characteristics of the fluids are studied and Nusselt Number correlations with Dean Number are developed.

The main objective in this work is to examine of the extent to which the proposed CFD-PBM modelling is able to capture the main features of the flow with heat transfer in helical pipes. The simulations were carried out as 3-D two-phase air-VG32 oil flow in a helical pipe based on the Eulerian–Eulerian description combined with Population Balance Modelling. The PBM model has been used to account for the non-uniform bubble size distribution in air-VG32 oil mixture. The geometry of helical pipe we have created using Autodesk inventor. The mesh element is used for the simulation is tetrahedral with prism layers. Total numbers of elements are 1.0 million.

2.1 Methodology

Two fluid model is most widely used to simulate the two phase flows. Euler/Euler volume-averaged continuity and momentum transport equations are written for each phase without any aggregation and breakage as:

Continuity:

$$\frac{\partial}{\partial t}\left(\rho_l \varepsilon_l\right)+\nabla.\left(\rho_l \varepsilon_l \mathbf{u}_l\right)=0 \text{, for liquid phase,} \tag{1}$$

$$\frac{\partial}{\partial t}\left(\rho_g \varepsilon_g\right)+\nabla.\left(\rho_g \varepsilon_g \mathbf{u}_g\right)=0 \text{, for gas phase,} \tag{2}$$

Momentum:

$$\frac{\partial}{\partial t}\left(\rho_k \varepsilon_k \mathbf{u}_k\right)+\nabla.\left(\rho_k \varepsilon_k \mathbf{u}_k \mathbf{u}_k\right)=-\varepsilon_k \nabla p+\varepsilon_k \rho_k \mathbf{g}+\sum \mathbf{F}_{km}+\varepsilon_k \mu_k \nabla^2 \mathbf{u} \quad (k,m = l,g). \quad (3)$$

where t is time, ρ_l and ρ_g is the density of liquid and density of gas, and $\mathbf{u}_l$, $\mathbf{u}_g$ is the liquid velocity and gas velocity respectively, ε_l is volume fraction of liquid and ε_g is volume fraction of gas.

2.2 Model geometry and Boundary conditions

Boundary conditions are specified for each boundary face. Each boundary control volume equation is ultimately closed when each boundary condition specification for each boundary face has been implemented. In general, boundary conditions can affect the control volume closure in any one of the two ways: Flux Discretised Boundary Conditions: The flow of the conserved quantity through each of the four boundary integration point surfaces is evaluated using the information provided by the boundary condition specification and control volume values within the element local to the face. These flow estimates are inserted into each of the four boundary control volumes sharing the boundary face. Boundary conditions implemented in this way include: walls, inlets, outlets and symmetry planes.

The pipe has an inner diameter d, the coil diameter is dc, and the distance between two adjacent turns, called pitch is hg. The results presented here are for a helically coiled pipe with 20 mm of inner pipe diameter, 300 mm coil diameter and 60mm pitch.

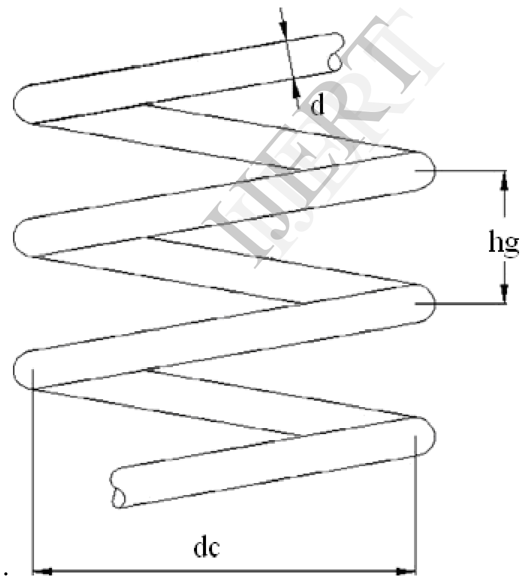

Figure 2. 1. The schematic of a helical pipe

Figure 2. 2 shows the geometry consider for the CFD model. This geometry we have created using Autodesk inventor. Figure 2.3 shows the mesh creation for CFD simulation. The mesh element is used for the simulation is tetrahedral with prism layers. Total numbers of elements are 1.0 million.

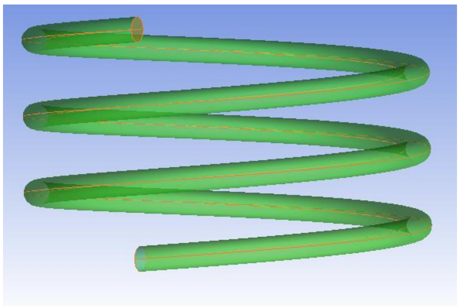

Figure 2. 2. Geometry considered for CFD simulations

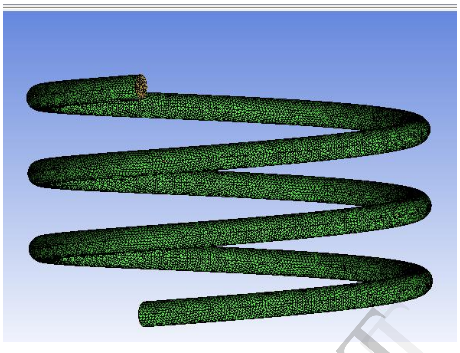

Figure .2. 3. Mesh generation for CFD simulations

2.3 Simulation Scheme

The simulations were carried out as 3-D two-phase air-VG32 oil flow in a helical pipe based on the Eulerian-Eulerian description by using CFX 13 combined with Population Balance Modelling. VG32 oil was considered as the continuous phase, and air was considered as the dispersed phase. Conservation equations are discretised using finite volume technique, high resolution scheme was used for all equations. The effect of gravitational force is applied in this analysis. The PBM model has been used to account for the non-uniform bubble size distribution in air-VG32 oil mixture. In the present study, bubbles are equally divided into 5 classes. Minimum and maximum of the bubbles diameter are 0.06 and 0.14 mm respectively. Several simulations were carried out using progressively larger number of grid points, until practically no change in the gas volume fraction and liquid velocity profiles was observed beyond a finite number of grids. At the pipe inlet, uniform gas and liquid velocities, temperature, turbulence intensity and average volume fractions have been specified; a relative average static pressure of zero was specified at the pipe outlet. No slip boundary conditions were used at wall. Average volume fraction and uniform liquid velocity profile are specified for initiating the numerical solution. Convergence criterion used was 1.0E−5 for all of the equations. In this analysis, a superficial velocity of 2.0 ms^{-1} for each of the phases and 20% volume fraction of air at the inlet are specified. For the heat transfer cases, hot fluid (air–VG32 oil mixture) at 360K enters at the top of the coil. The pipe wall temperature is maintained constant at 300K. Figure 3 shows the inlet and outlet boundary conditions specified locations.

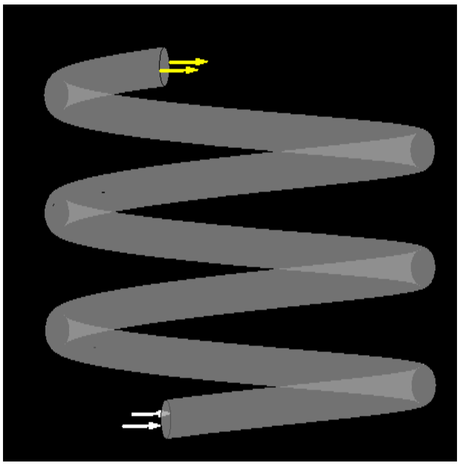

Figure 2.3: Inlet and outlet boundary conditions specifications

3. Results and Discussions

This validation process consist of CFD result of the current project and the Experimental result of the author Jayakumar et al.[7]. From these values validation can be done.

Table 3.1 Gives the comparison between CFD and experimental result.

Sl no	Experimental values of [1] for pressure drop in Kpa/m	CFD results For pressure drop in Kpa/m	difference	% error
1	2.50	2.27	0.092	9.2

3.1 Variation of air volume fraction.

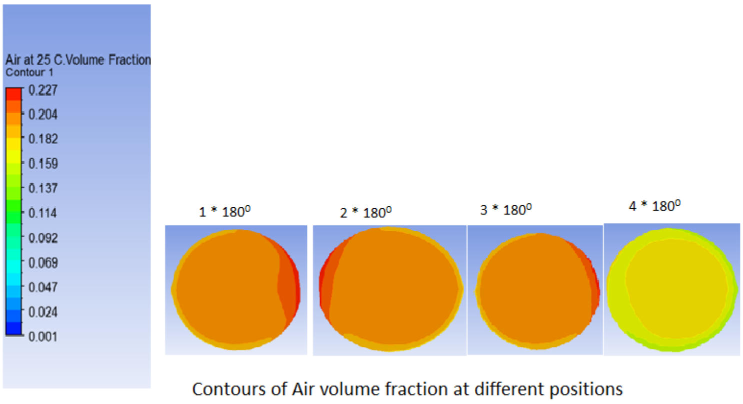

Fig.3.1 Contour plots of air volume fraction at different positions, CFD – PBM model

The contour plots of air volume fraction are shown in Fig.3.1 with PBM. From the counter plots we observed that the air volume fraction becomes maximum at outer side of the pipe while further moving volume fraction becomes high at the centre of the pipe.

From above fig 1*180°, in this section of helical pipe the air volume fraction becomes high at inner section of the pipe that is around 0.227 while remaining section of the pipe have the air volume fraction around 0.204. Similarly at 2*180° air volume fraction becomes maximum at outer side of the pipe .Thus at the outer end of pipe air volume becomes less at outer side of the pipe while maximum at centre of the pipe shown in 4*180° in this minimum air volume is around 0.158.

3.2 Variation of VG 32 oil pressure.

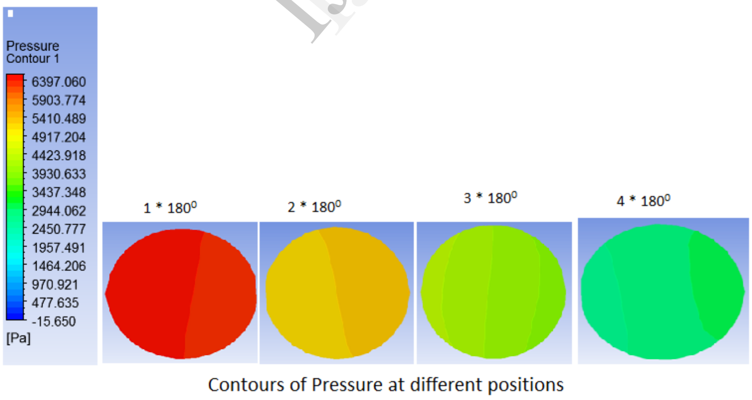

Fig.3.2 Contour plots of pressure at different positions, CFD – PBM model

The contour plots of pressure at different planes presented in Fig. 3.2 with PBM Centrifugal forces leads to the higher concentration region of VG32 oil at the outer side of the pipe wall. The acceleration forces acting on the fluid flow on the pipe lead to a creation of high pressure region at outer side of the pipe wall . In other words, raising the pressure on the outer side of the wall caused the buoyancy forces, therefore air bubbles tend to leave it. From fig 3.2 at 1*180° inlet section of the pipe where pressure becomes maximum that is around 6397.64 pa. similarly at outer section of the pipe at 4*180° pressure becomes lower that is around 4423.98 pa thus pressure drop of the VG 32 oil is 2273.66 pa.

3.3 Variation of air velocity.

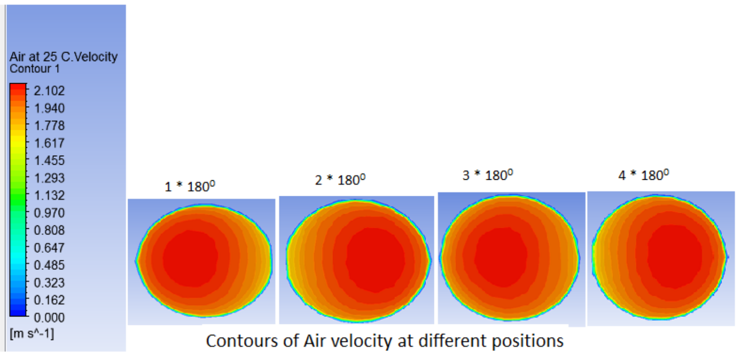

Fig.3.3 Contour plots of air velocity at different positions, CFD – PBM model.

The contour plots of air velocity are shown in Fig.3.3 with PBM . Because of centrifugal forces, high velocity region is observed at the outer side of the pipe, whereas in the case of flow in a straight pipe, high velocity region is observed at the centre of the pipe. The location of high velocity region shifted slightly down in successive turns of pipe, and the band of very 6high air velocity (dark red band) is slightly wider also. Therefore, the flow is unsymmetrical about the horizontal plane of the coil.

3.4 Variation of VG 32 oil temperature.

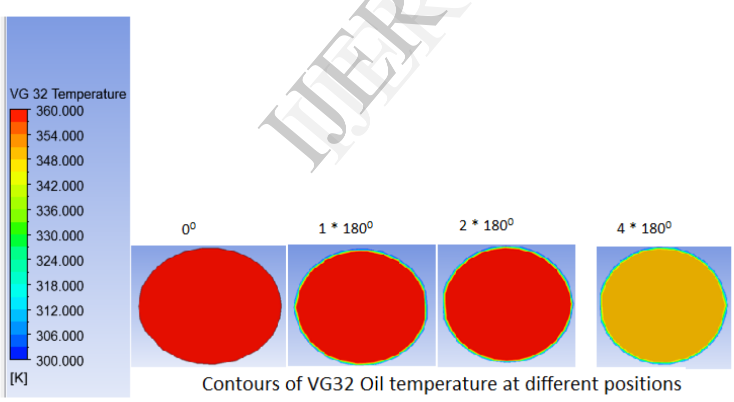

fig 3.4 Contour plots of temperature of VG32 Oil at different position

The contour plots of VG32 oil temperature are shown in Fig. 3.4. It can be seen that the temperature decrease as fluid flows along the pipe. The difference in temperature at the inlet of the pipes walls is higher than outer. In these contour plots, visible differences can be seen between simulation with PBM model. In an overall view the CFD-PBM model has gives better results.

4.0 CONCLUSION

Air-VG32 oil two-phase flow with heat transfer through a helical pipe is simulated using computational fluid dynamic modelling combined with a population balance model. The bubble size distribution and coalescence and breakage of bubble groups handled through the population balance model (PBM). It was found that centrifugal forces cause creation of a high velocity region at the outer side of the helical pipe walls. Centrifugal forces leads to the higher volume fraction region of VG32 oil at the outer side of the helical pipe wall. In other words, increasing the pressure on the outer side of the wall causes buoyancy forces, and therefore the air bubbles tend to migrate away from the wall. Finally the CFD-PBM model has more abilities to capture the flow features.

References

[1] S.A. Berger, L.Talbot, , L.S. Yao, Flow in curved pipes, Ann Rev Fluid Mech. 15 (1983) 461-512.

[2] S. Vashisth, V.Kumar, K.D.P Nigam,. A Review on the Potential Applications of Curved Geometries in Process Industry, Ind. Eng. Chem. Res., 47 (10), pp 3291–3337(2008).

[3] P. Naphon, S. Wongwises, A review of flow and heat transfer characteristics in curved tubes,Renewable and Sustainable Energy Reviews 10 463–490(2006).

[4] T.J. Huttl and R. Friedrich, "Direct numerical simulation of turbulent flows in curved and helically coiled pipes", Computers and Fluids 30, 591 (2001).

[5] R. Friedrich, T.J. Huttl, M. Manhart and C. Wagner, "Direct numerical simulation of incompressible turbulent flows", Computers and Fluids 30, 555 (2001).

[6] J.S.Jayakumar, , S.M. Mahajani, J.C. Mandal, K.N. Iyer, P.K. Vijayan, CFD analysis of single-phase flows inside helically coiled tubes, Computers and Chemical Engineering 34 (4), pp. 430-446 (2010).

[7] J.S. Jayakumara, S.M. Mahajani, J.C. Mandal, Kannan N. Iyer, P.K. Vijayan, Thermal hydraulic characteristics of air–VG32 oil two-phase flows in helical pipes, chem. eng. res. des. 88 501–512 (2010).

[8] H. Luo, H. Svendsen, Theoretical model for drop and bubble break-up in turbulent dispersions, AIChE J. 42 1225–1233 (1996).

[9] M.J. Prince,H.W. Blanch, Bubble coalescence and break-up in air sparged bubble columns, AIChE J. 36 1485– 1499 (1990).

[10] M. Ishii, N. Zuber, Drag coefficient and relative velocity in bubbly, droplet or particulate flows, AIChE J. 25 843–855(1979).

[11] Y. Sato, K.Sekoguchi, Liquid velocity distribution in two-phase bubble flow, Int. J. Multiphase Flow, 2 79(1975).

[12] G.Kasturi, and J.B., Stepanek, Two phase flow. I. Pressure drop and void fraction measurements in cocurrent gas–liquid flow in a coil. Chem Eng Sci, 27: 1871–1880(1972).