International Journal of Chemical, Materials and Biomolecular Sciences
ISSN: 2415-6620
Vol:7, No:1, 2013

CFD Simulation and Validation of Flow Pattern Transition Boundaries during Moderately Viscous Oil-Water Two-Phase Flow through Horizontal Pipeline

Anand B. Desamala, Anjali Dasari, Vinayak Vijayan, Bharath K. Goshika, Ashok K. Dasmahapatra, and Tapas K. Mandal

Abstract**—**In the present study, computational fluid dynamics (CFD) simulation has been executed to investigate the transition boundaries of different flow patterns for moderately viscous oil-water (viscosity ratio 107, density ratio 0.89 and interfacial tension of 0.032 N/m.) two-phase flow through a horizontal pipeline with internal diameter and length of 0.025 m and 7.16 m respectively. Volume of Fluid (VOF) approach including effect of surface tension has been employed to predict the flow pattern. Geometry and meshing of the present problem has been drawn using GAMBIT and ANSYS FLUENT has been used for simulation. A total of 47037 quadrilateral elements are chosen for the geometry of horizontal pipeline. The computation has been performed by assuming unsteady flow, immiscible liquid pair, constant liquid properties, co-axial flow and a T-junction as entry section. The simulation correctly predicts the transition boundaries of wavy stratified to stratified mixed flow. Other transition boundaries are yet to be simulated. Simulated data has been validated with our own experimental results.

Keywords**—**CFD simulation, flow pattern transition, moderately viscous oil-water flow, prediction of flow transition boundary, VOF technique.

I. Introduction

WATER lubrication technique is one of the most preferable economic ways to reduce the pressure drop in the transportation of high viscous oil through pipe network. Water lubrication reduces wall friction during the flow. In such flow, the phases (viz., oil and water) can arrange themselves in various patterns (viz., flow pattern), to minimize their total energy which is summation of potential, kinetic and surface (viz., oil-water interface) energy. This flow pattern greatly influences pressure drop and hold-up characteristics; heat and mass transfer characteristics and kinetics between two immiscible phases. Separated patterns (viz. smooth stratified, wavy stratified, mixed stratified and core annular flow) especially, annular flow is preferable from pressure drop point of view where as dispersion (viz. oil dispersed in water and water dispersed in oil) is important from the heat and mass transfer and kinetics point of view. Information about transition boundaries of different flow patterns helps in accurate designing of various unit operations and processes, which have motivated various researchers to investigate on predicting transition boundaries. For example, the transition boundary of stratified to dispersed flow [1], wavy stratified to stratified mixed flow [2], stratified to intermittent (viz. plug and slug) flow [3] has been predicted using various analytical models. Owing to its immense industrial importance, several researchers have also conducted experiments of oil-water flows in different pipe orientations and identified different flow patterns by visualization, imaging [4-5] and probe (intrusive and non intrustive) [6-7] techniques to get the experimental transition boundaries of the various flow patterns. To understand the physics behind it, some of the researchers have predicted the flow regimes by CFD simulations [8-11]. Among these Ko et al.[9] have simulated turbulent core flow using different turbulence models while Ghosh et al. [10] have simulated the core annular down flow. Recently Al-Yaari and Abu-Sharkh [11] have predicted the stratified oil-water flow through horizontal pipe using VOF techinque with RNG k-ε turbulance model.

Based on the lacuna of the past literature, the objective of the present study has been choosen. In the present study, computational fluid dynamics (CFD) simulation has been executed to investigate the transition between different flow patterns for moderately viscous oil-water (viscosity ratio 107, density ratio 0.89 and interfacial tension of 0.032 N/m.) two-phase flow through a horizontal pipeline with internal diameter and length of 0.025 m and 7.16 m respectively. Volume Of Fluid (VOF) approach including effect of surface tension [12] has been tried to adopt for prediction of flow regime transition boundaries. To validate the simulated result, experiments of oil-water flow through horizontal pipeline is carried out for a wide range superficial velocities of both oil and water covering all the flow patterns.

II. Model Development

Fig. 1 depicts the geometry and flow passage considered for the present computational modeling. The geometry consists of a horizontal pipeline with internal diameter and length of 0.025 m and 7.16 m respectively.

All Authors are with the Department of Chemical Engineering, Indian Institute of Technology Guwahati, Guwahati – 781039, Assam, India (phone: + 91-3612582271; fax: + 91-3612582291; Tapas K Mandal e-mail: tapasche@iitg.ernet.in, Ashok K Dasmahapatra e-mail: akdm@iitg.ernet.in).

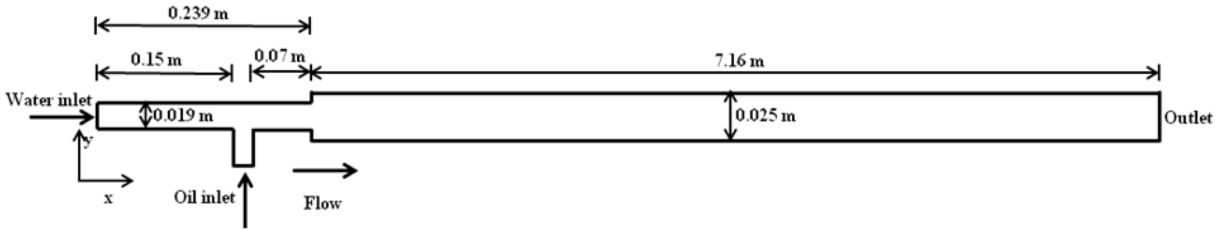


Fig. 1 Schematic of model geometry

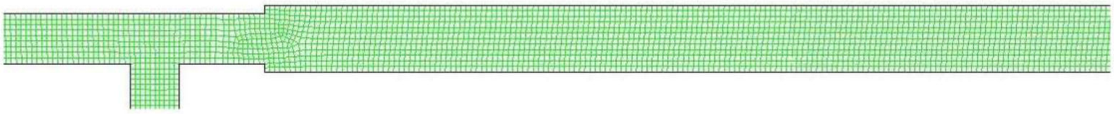


Fig. 2 Mesh geometry of the entry section and pipe line

Oil and water are introduced into the pipe through a T-junction at the entry section where water and oil enter into the pipe from the horizontal and vertical directions, respectively. The geometry consists of four sections -water inlet, oil inlet, outlet of the pipe and the test section. The meshing of the present model has been done using GAMBIT. Fig. 2 depicts the meshed geometry. The mesh consists of 44330 and 2707 quadrilateral mesh elements for pipeline and entry section respectively. Quadrilateral mesh geometry is selected for accounting surface tension effect more accurately.

The CFD software package of ANSYS FLUENT^{TM} has been used for simulation. The computation has been performed by assuming unsteady flow, immiscible liquid pair, constant liquid properties and co-axial flow in the pipe and a T-junction as entry section. In the present model the two fluids share a well defined interface, the Volume of Fluid (VOF) approach for two phase modeling has been selected in Fluent. VOF solves a single set of momentum equations which is shared by both the fluids. The details of the governing equations and the treatment of the interface are obtained from Fluent user guide [12].

A. Governing Equations

Continuity:

$$\frac{\partial(\rho)}{\partial t} + \nabla \cdot (\rho U) = \sum_q S_q \tag{1}$$

Where, ρ, *U*, *t*, *S* are density, velocity, time and mass source, respectively. In the present case *S* is zero.

Momentum:

The single momentum equation is solved in the computational domain and the resulting velocity field is shared amongst the phases. The general momentum equation can be written as:

$$\frac{\partial(\rho U)}{\partial t} + \nabla \cdot (\rho U \cdot U) = -\nabla P + \nabla \cdot [\mu(\nabla U + \nabla U^T)] + (\rho g) + F \tag{2}$$

Where *P*, *g*, *F*, *μ* are pressure in the flow field, acceleration due to gravity, body force acting on the system and viscosity of the flowing fluid, respectively.

B. Secondary Phase Tracking

The VOF method is capable of tracking the volume fraction of each liquid in each computational cell. In each control volume, the fraction of all the phases sum up to unity. The model enforces the fact that all variables and physical properties be shared by each phase. If the volume fraction of the phases is known in each cell, the physical properties can be represented as volume averaged values. Thus, from the volume fraction values and the volume averaged properties, we can detect the presence of either of the phases in a particular cell [12]. In other words, if the volume fraction of q^{th} fluid in the cell is denoted as α_q, the following three conditions are possible:

- $\alpha_q = 0$: the cell does not contain fluid q.
- $\alpha_q = 1$: the cell is occupied solely by fluid q.
- $0 < \alpha_q < 1$: the cell contains the interface between the q^{th} fluid and one or more other fluids.

Depending on the local value of α_q, the appropriate properties and variables are calculated in each control volume within the computational domain. For example, the density and viscosity used in **"(1),"** and **"(2)"** are estimated as:

$$\rho = \sum_1^p \rho_q \alpha_q \tag{3}$$

$$\mu = \sum_1^p \alpha_q \mu_q \tag{4}$$

A separate continuity equation for α_q is considered as follows:

$$\frac{\partial \alpha_q}{\partial t} + (U_q \cdot \Delta) = S_{\alpha q} \tag{5}$$

For each of the cells the following relationship is also valid:

$$\sum_{1}^{p} \alpha_q = 1 \tag{6}$$

Where "p" is the number of phases. For the present two phase flow, $p = 2$.

C. Interface Treatment

In order to construct the interface between the fluids, VOF utilizes a piecewise-linear approach. It assumes that the interface between two fluids is characterized by a linear slope within each cell. The scheme uses this linear shape for the calculation of the advection of fluid through the cell faces. In the first step of interface reconstruction, based on the information of the volume fraction and the derivatives in each cell, the position of the linear interface relative to the centre of each partially filled cell is calculated. After that, the advection of fluid through each face is calculated using the computed linear interface representation and information about the normal and tangential velocity distribution on the face. Finally the volume flux in each cell is obtained by using the balance of fluxes calculated during the previous step [12].

D. Surface tension and Wall adhesion

The VOF method incorporated the effect of surface tension along the interface between the phases. In the present case, the contact angle between the phases and the wall is specified in the model and the surface tension coefficient is taken as constant.

The surface tension model uses the continuum surface force (CSF) model [13]. In this model, the addition of surface tension to the VOF calculation results in a source term in the momentum equation where the pressure drop across the surface can be obtained from "surface tension coefficient σ" and the surface curvature which is measured by two radii in the orthogonal directions "R_1" and "R_2" using Young-Laplace equation [12] as follows,

$$P_2 - P_1 = \sigma\left(\frac{1}{R_1} + \frac{1}{R_2}\right) \tag{7}$$

P_1 and P_2 in **"(7),"** are the pressures in the two fluids on either sides of the interface.

E. Initial and Boundary Conditions

1) Initial Condition

In all the simulations, the flow is initiated from water inlet and the pipe is initially filled with water.

2) Initial Boundary Condition

The water velocity and oil velocity are specified at its inlet to the T-junction and uniform velocity distribution is assumed at the inlet for both fluids.

At x = 0 and y = 0, U_y=0 and U_x=U_{water}

At x=0.15 m and y= -0.0595 m, U_y= U_{oil} and U_x=0

3) Wall Boundary Conditions

A stationary no-slip boundary condition is imposed on the wall of the pipe. The contact angle with magnitude 8.5^o between oil and pipe material in water medium is provided.

4) Outlet Boundary Conditions

At the outlet, pressure outlet boundary is used and diffusion fluxes for the variables in the exit direction are set to zero.

5) Boundary Conditions

There are three sections bounding the calculation domain: the inlet boundary, the wall boundary and the outlet boundary. Wide range of velocities for oil and water are introduced at the inlet of their sections. The outlet boundary condition **is** set up as a pressure outlet boundary. No slip condition is used to model liquid velocity at wall.

F. Discretization Method

Flow patterns of two phase flow are dynamic in nature. So, its variation with time and space has been considered during the simulation. A transient simulation has been carried out with a time step of 0.001 s. In this simulation continuity equations are discretized by PRESTO [14] while momentum equations are discretized by first order upwind method. PISO algorithm [15] is used for pressure velocity coupling.

G. Solution Methodology

All the above mentioned equations are solved in Fluent by VOF method which is nothing but a control volume based technique. The equations are first converted into algebraic equations which are then solved numerically. Unsteady State and pressure based solver **is** employed in this case. The typical time steps used in this computation **is** around 0.001 s. The equations are integrated over the control volume and the parameters (pressure and velocity) are conserved by solving the momentum and continuity equations using the Piso pressure-velocity coupling scheme. The continuity equations are discretized using Presto algorithm while the momentum equations utilize a first order upwind discretization scheme. For the interface, VOF uses the Geometric Reconstruct interpolation scheme.

III. Experimentation

A. Identification of Flow Pattern Transition Boundaries

In order to validate the simulation results, experiments have been carried out with lube oil (890 kg/m^3, 0.107 Pa.s, 0.024 N/m) and water (1000 kg/m^3, 0.001 Pa.s, 0.072 N/m) as test fluids with an interfacial tension of 0.032 N/m in horizontal pipeline as shown in Fig. 3. One centrifugal pump (P1) and one gear pump (GP) have been used to pump water and oil respectively to the entry section of the test rig. Water flow rates have been measured with the help of pre-calibrated rotameters (RM1 and RM2). Oil flow rate has been measured by volumetric technique at the exit section.

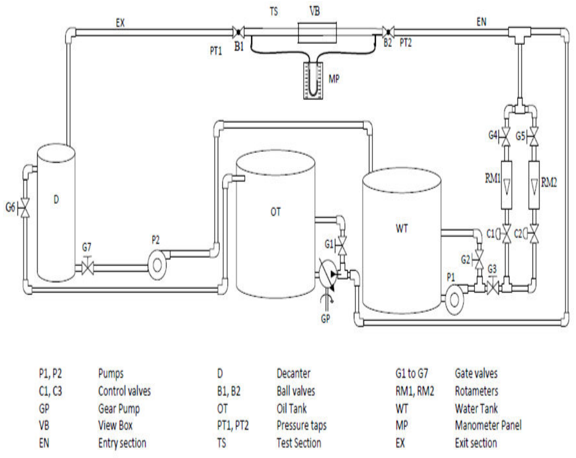


Fig. 3 Schematic of experimental setup

Experiments have been performed by increasing oil velocity gradually at a constant water velocity. In steady state, a snap shot has been taken using a digital camera (DSC-HX100V, Sony) at the test section to identify the flow pattern. This experiment has been repeated thrice to check the reproducibility of the experimental results. After completion of a set of experiment, the water velocity is changed to the next higher value and the readings are repeated as earlier. The oil superficial velocity (U_{SO}) has been varied from 0.015 m/s to 1.25 m/s and the water superficial velocity (U_{SW}) from 0.1 to 1.1 m/s to cover the entire range of flow patterns.

B. Contact angle Measurement

The contact angle between lube oil and solid Perspex wall has been measured in water environment because it is needed in simulation. Phase arrangement in contact angle measurement is shown in Fig. 4. The contact angle is measured experimentally and the value of the contact angle is incorporated in the simulation. The pipe material used in the present study is made up of Perspex. So Perspex surface is used for contact angle measurement. Prior to measurement the perspex surface is equilibrated with water. Then an oil droplet is carefully placed on to the surface, which is already immersed in the water medium. An image of the oil droplet in this arrangement is taken with aid of digital camera [DSC-HX100V, Sony]. These images are analyzed using the software Adobe Acrobat 9.0. After analyzing the image the contact angle is found to be $8.5^o \pm 2^o$. The average value (8.5^o) of this contact angle is used in the present simulation.

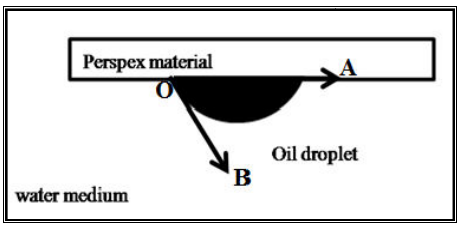


Fig. 4 Illustration of the phase arrangement for the contact angle measurement

IV. Results and Discussion

A. Experimental Results

Experimentally identified flow patterns are shown as flow patterns map in Fig. 5. X-axis and Y-axis of this plot represent superficial velocity of oil and water respectively. Six flow patterns namely plug (P), slug (S), wavy stratified (SW), stratified mixed (SM), Dispersion of oil in water ($D_{O/W}$) and dispersion of water in oil ($D_{W/O}$) have been identified in the present work and these are shown in the Fig. 5 by different symbols.

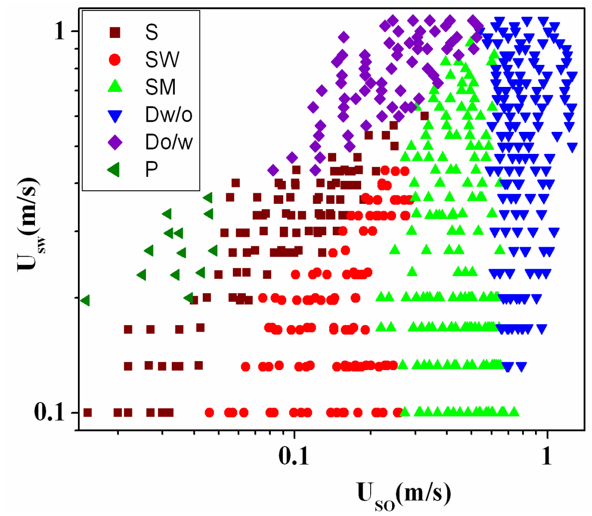
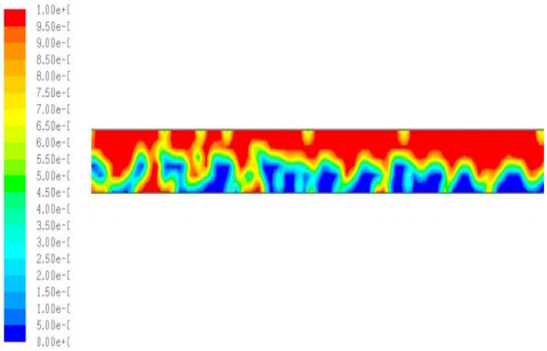


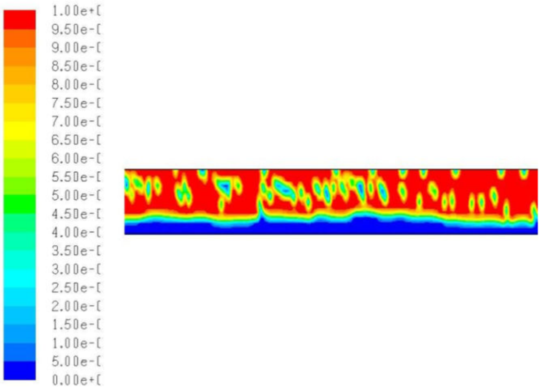
Fig. 5 Experimental flow pattern map
(◄- plug flow (P) , ■- slug flow (S), •-wavy stratified flow (SW), ▲-Stratified mixed flow (SM),▼-water dispersion in oil flow ($D_{w/o}$),♦ -oil dispersion in water flow($D_{o/w}$))

B. Study of Grid Independency

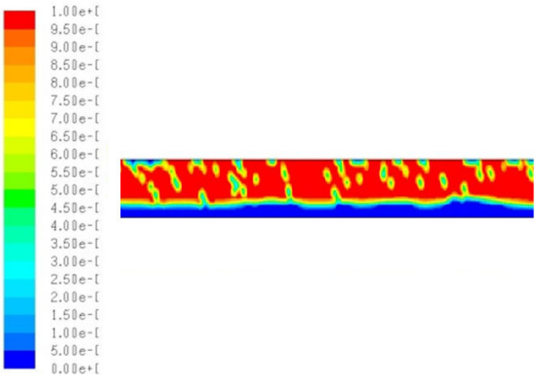
The computational grids of 14,732, 47,037 and 63,191cells have been tested for the grid independent study to find out the optimum size of the mesh to be used for the simulation. For this, the simulation has been done for one case study using water superficial velocity, U_{sw} = 0.23 m/s and oil superficial velocity, U_{so} = 0.29 m/s. The oil volume fraction contours with these three computational grids are presented in figure 6. In these figures blue color represents the presence of water while the red color represents the oil. Fig. 6 depicts that the system with 14,732 cells was not able to predict the stratified flow pattern. So, the extra numbers of cells are needed to be examined. System with 47,037 and 63,191 cells almost predict the oil volume fraction with clear interface. Therefore, based on the oil volume fraction contour results, 47,037 cells are selected as the optimum number of cells to be used in the simulation.



(a) System with 14,732 Cells



(b) System with 47,037 Cells

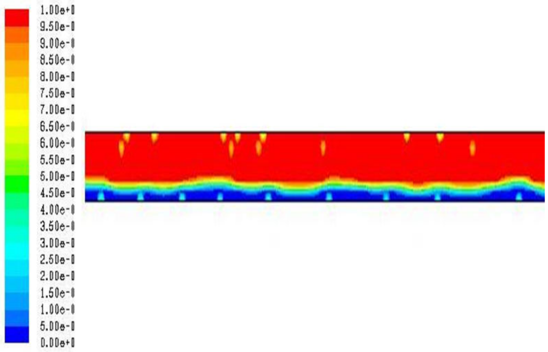


(c) System with 63,191 Cells

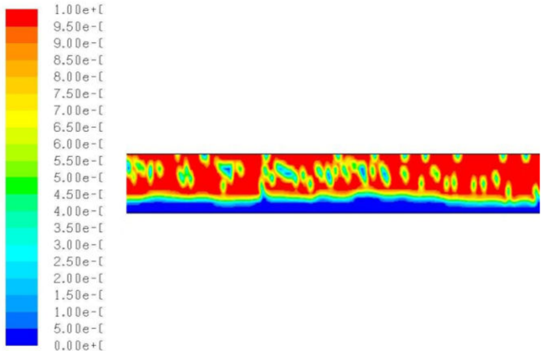
Fig. 6 Oil volume fraction contours for grid independent study at U_{sw} = 0.23 m/s, U_{so}= 0.29 m/s

C. Simulation Results

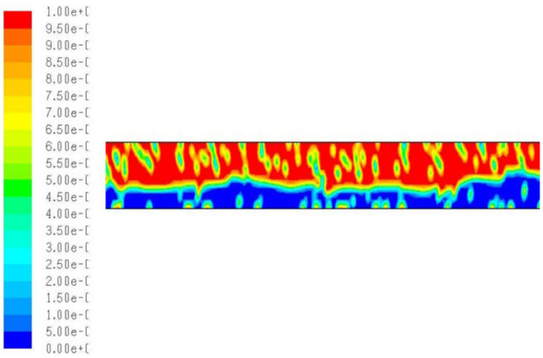
Simulations have been performed for prediction of different flow pattern transition boundaries observed in the experimental results. The experimental transition boundaries are (1) plug to slug, (2) slug to wavy stratified, (3) wavy stratified to stratified mixed, (4) stratified mixed to water dispersed in oil, (5) slug to oil dispersed in water, (6) wavy stratified to oil dispersed in water and (7) stratified mixed to oil dispersed in water. Simulation has been started with 44,330 grids because it gives the best solution for the wavy stratified



(a) U_{sw} = 0.1 m/s, U_{so}= 0.275 m/s



(b) U_{sw} = 0.23 m/s, U_{so}= 0.29 m/s



(c) U_{sw} = 0. 3 m/s, U_{so}= 0.3 m/s

Fig. 7 Simulated results of Stratified Mixed flow pattern at different velocity of both the phases

- to stratified mixed transition boundary as mentioned in above section. Simulation has been run at various flow rates of both the phases to get the various data points along this transition boundary. The three of them have been shown in Fig. 7 as the sample results at three different flow rates. Those are (1) U_{sw} = 0.1 m/s, U_{so}= 0.275 m/s; (2) U_{sw}= 0.23 m/s, U_{so}= 0.29 m/s; and (3) U_{sw}= 0.3 m/s, U_{so}= 0.3 m/s. All figures represent the contour of oil volume fraction for different superficial velocities of both the phases. One representative photograph of Stratified Mixed flow pattern at U_{sw} = 0.1 m/s, U_{so}= 0.275 m/s obtained experimentally is also shown in Fig. 8. Different data points of transition between wavy stratified and stratified mixed flow pattern obtained from the simulations are placed on the experimental flow pattern map and connected with solid pink line as shown in Fig. 9. The figure shows good agreement with the experimental result. Other transition boundaries may be obtained in similar ways which are not done yet. Therefore more effort should be put to get all other transition boundaries.

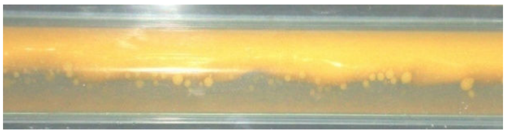


Fig. 8 One representative photograph of stratified mixed flow pattern at U_{sw} = 0.1 m/s,U_{so}= 0.275 m/s obtained from the experiment

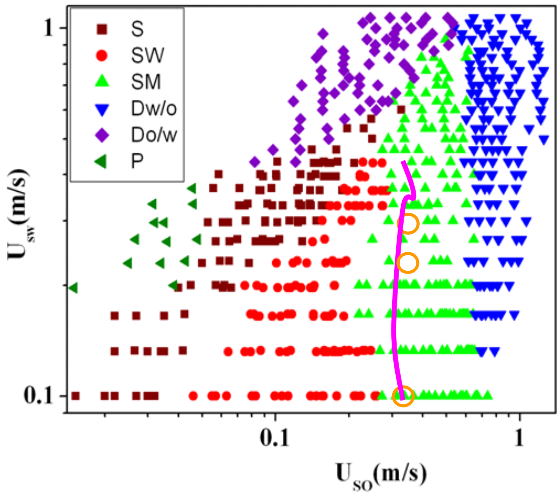


Fig. 9 Validation between experimental transition boundary and simulated transition boundary indicated by solid line of wavy stratified to stratified mixed flow pattern. Circled data points are the simulated and superimposed on experimental flow pattern map.

V. Conclusion

The attempts have been made to predict the flow pattern transition boundaries of moderately viscous oil and water two-phase flow through horizontal pipe. CFD simulation has been done with the help of ANSYS FLUENT^{TM} software. Grid independent study has been done and system with 47037 cells is selected as optimum number of cells for simulation. Transition boundary of wavy stratified to stratified mixed flow has been predicted well using VOF technique. The simulated result is also validated with the experimental results. A good agreement has been observed for this transition boundary. Other transition boundaries like wavy stratified to slug, mixed stratified to both the dispersion either oil in water and water in oil have not been predicted yet. The simulation is going on and it will take time to get the results.

References

[1] N. Brauner, "The prediction of dispersed flows boundaries in liquid-liquid and gas-liquid systems," *Int. J. Multiphase Flow*, vol. 27, pp. 885–910, May 2001.

[2] N. Brauner and D. Moalem Maron, "Analysis of stratified/non-stratified transitional boundaries in inclined gas-liquid flows," *Int. J. Multiphase Flow*, vol. 18, pp. 541–557, July 1992.

[3] A. Soleimani and T. J. Hanratty, "Critical liquid flows for the transition from the pseudo-slug and stratified patterns to slug flow," *Int. J. Multiphase Flow*, vol. 29, pp. 51–67, January 2003.

[4] T. W .F. Russell, G. W. Hodgson and G. W. Govier, "Horizontal pipeline flow of mixtures of oil and water," *Can. J. Chem. Eng.*, vol. 37, pp. 9–17, February 1959.

[5] J. L. Trallero,C. Sarica and J. P. Brill, "A study of oil/water flow patterns in pipes," *SPE Prod. Facil. 36609*, pp. 165–172, August 1997.

[6] P. Angeli, and G. F. Hewitt, "Flow structure in horizontal oil-water flow," *Int. J. Multiphase Flow*, vol. 26, pp. 1117–1140, July 2000.

[7] D. P. Chakrabarti, G. Das and P. K. Das, "Identification of stratified liquid–liquid flow through horizontal pipes by a non-intrusive optical probe," *Chem.l Eng. Sci.*, vol. 62, pp. 1861–1876, April 2007.

[8] Sumana Ghosh, Gargi Das and Prasanta Kumar Das, "Simulation of core annular in return bends–A comprehensive CFD study," *Chem. Eng. Res. Des.*, vol. 89, pp. 2244–2253, November 2011.

[9] T. Ko, H. G. Choi, R. Bai and D.D. Joseph, "Finite element method simulation of turbulent wavy core-annular flows using a k-w turbulence model method," *Int. J. Multiphase Flow*, vol. 28, pp. 1205–1222, July 2002.

[10] Sumana Ghosh, Gargi Das and Prasanta Kumar Das, "Simulation of core annular downflow through CFD–A comprehensive study," *Chem.l Eng. Process*, vol. 49, pp. 1222–1228, November 2010.

[11] Mohammed A. Al-Yaari and Basel F. Abu-Sharkh, "CFD prediction of stratified oil-water flow in a horizontal pipe," *Asian Transactions on Engineering*, vol. 01, Issue 05, pp. 68–75, November 2011.

[12] Fluent 6.3 User's Guide, Fluent Inc., Lebanon, USA, 2006.

[13] J. U. Brackbill, D. B. Kothe and C. Zemach, "A Continuum Method for Modeling Surface Tension," *J. Comput. Phys.*, vol. 100, pp. 335–354, 1992.

[14] S. V. Patankar, *Numerical Heat Transfer and Flud Flow*, Hemisphere, Washington, DC., 1980.

[15] R. I. Issa, "Solution of the implicitly discretized fluid flow equations by operator splitting", *J. Comput. Phys.*, vol. 62, pp. 40-65, 1986.