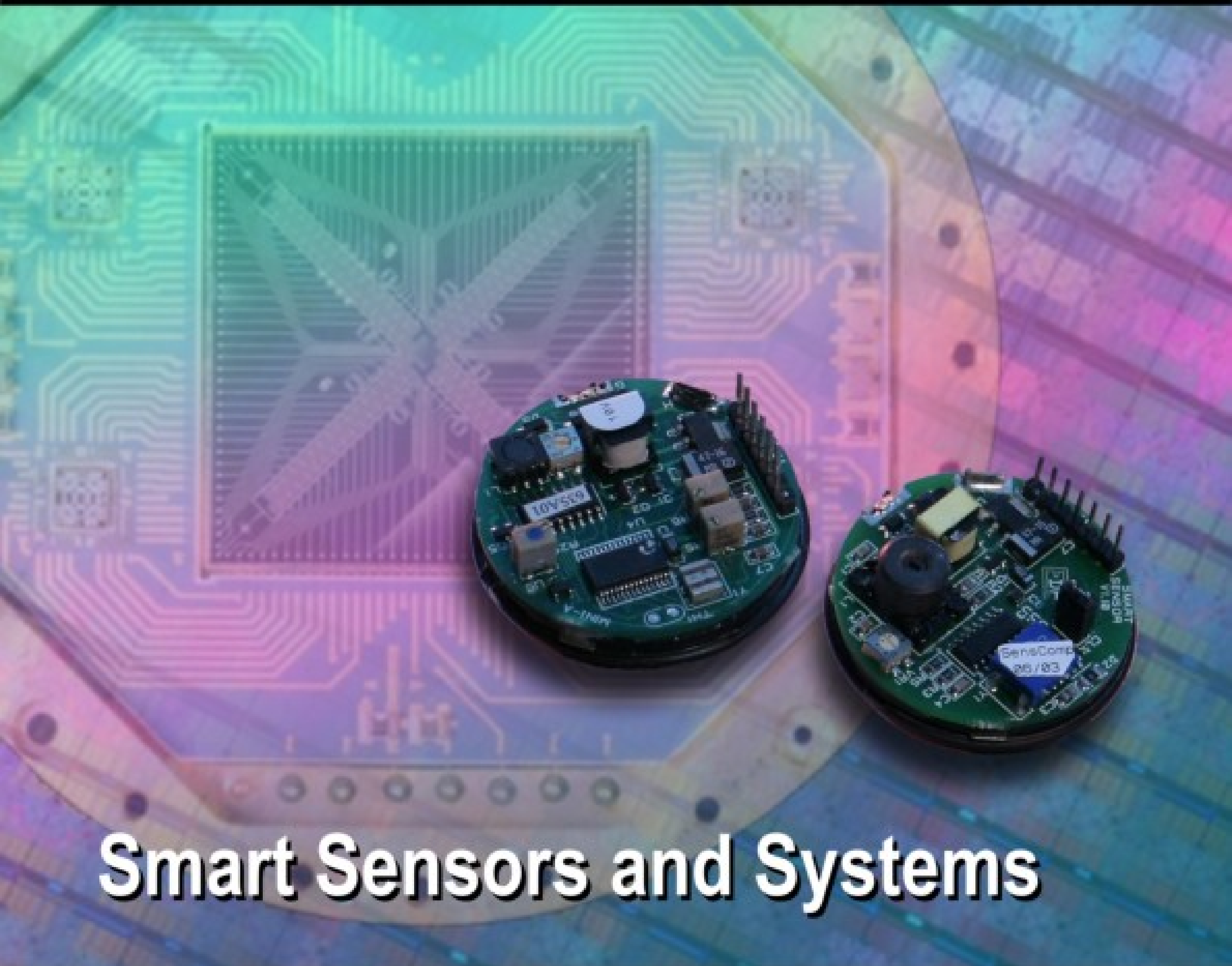
ISSN 1726-5479

SENSORS & TRANSDUCERS

3/09 vol. 102



Smart Sensors and Systems

International Frequency Sensor Association Publishing

IFSA

Sensors & Transducers

Volume 102, Issue 3
March 2009

www.sensorsportal.com

ISSN 1726-5479

Sensors & Transducers Journal (ISSN 1726-5479) is a peer review international journal published monthly online by International Frequency Sensor Association (IFSA).
Available in electronic and CD-ROM.

Sensors & Transducers Journal

Contents

Volume 102
Issue 3
March 2009

www.sensorsportal.com

ISSN 1726-5479

Research Articles

Authors are encouraged to submit article in MS Word (doc) and Acrobat (pdf) formats by e-mail: editor@sensorsportal.com
Please visit journal's webpage with preparation instructions: http://www.sensorsportal.com/HTML/DIGEST/Submition.htm

International Frequency Sensor Association (IFSA).

Sensors & Transducers Journal, Vol. 102, Issue 3, March 2009, pp. 94-104

Sensors & Transducers

ISSN 1726-5479

http://www.sensorsportal.com

Experimental and Computational Study of Two-Phase (Air–Palm Oil) Flow through Pipe and Control Valve in Series

Arivazhagan M., Pugalenthi, Krishna Karthik K., Rani Hemamalini, [*]Sundaram S.

Department of Chemical Engineering, National Institute of Technology, Tiruchirapalli-620015, India

[*]Department of electronics and instrumentation, SASTRA, Thanjavur, India

Tel.: +91431-2503111, +919487412478

E-mail: ariva@nitt.edu

Received: 16 October 2008 /Accepted: 24 March 2009 /Published: 31 March 2009

Abstract: The contact of two or more immiscible liquids is encountered widely in the chemical and petroleum industries. Studies on operating characteristics of control valves with two phase flow have not been given much attention in the literature despite its industrial importance during design and selection as well as plant operations .The present work attempts to study experimentally the effect of two phase flow on pressure drop across pipe and control valve in series and compare with simulated results. Two-phase computational fluid dynamics (CFD) calculations, using commercial CFD package FLUENT 6.2.16, were employed to calculate the simulated the pressure drop in Air–Palm oil flow in pipes and control valves. The Air flow rate varied from 25 to100 l/h flow rate. For constant valve position and Air flow rate, the Palm oil flow rate was varied from 50 to 150 l/h. The numerical results were validated against experimental data. The prediction of the pressure drop characteristics in pipe and valve were within an average error of about ± 3 %. A comparison of experimental and computed profiles was found to be in good agreement. *Copyright © 2009 IFSA.*

Keywords: Air-Palm oil, Two phase flow, Pressure drop, CFD modeling, Fluent

1. Introduction

Simultaneous gas and liquid flow occurs in many types of industrial equipments, such as high pressure boilers, condensers, thermal hydraulic circuits of nuclear power stations, refrigeration equipments, evaporators and many other parts of chemical and process plants. Due to the extreme conditions of the large – scale industrial plant operations (for example high pressure and consequent high capital cost), the fundamental research on multiphase flows has been directed towards the gas-liquid studies at low pressure.

The dispersion of the two phases in the flow channel depends on the flow rate of each phase, the flow properties and the channel geometry. The particular way the two phase are dispersed is termed the flow pattern or flow regimes and varies from one flow to another. These variations often make the two phases flow predictions, for example pressure drop and heat transfer characteristics difficult to obtain and in most cases not as reliable as the corresponding single phase ones. However the design of many industrial components necessities the reasonable estimation of these two phase parameters.

The studies in two-phase flow through pipes have been conducted for the past 60 years. The first detailed study on two-phase flow was carried out by Lockhart and Martinelli [18] in 1949. Design of pipe line for the simultaneous flow of oil and gas was discussed by Baker [1] and Hoogendoorn [2] has studied the gas-liquid flow in horizontal pipes. The two phase slug flow in horizontal and inclined tubes was discussed by Vermeulenand Ryan [3]. Beretta et al. [4] has studied pressure drop for horizontal oil-palm oil flow in small diameter tubes. In another study Awwad et al. [5] analyzed flow patterns and pressure drop with air-palm oil in horizontal helicoidal pipes .A comparison of existing theories on two phase flow was analyzed by Kordbyban [6]. Pressure gradients due to friction for the two-phase mixtures in smooth tubes and channels were studied by Chisholm [7]. They are successors in the field of two- phase flow and the studies were concentrated on developing the flow pattern model for horizontal, inclined and vertical flow in pipes.

Oliemans and Ooms [8] gave a semi-empirical model for the core- annular flow of oil and palm oil through a pipeline. Core-annular flow of two immiscible fluids through pipeline was studied by Bai et al. [9] and they used an oil with viscosity of 600 times the palm oil viscosity and found that the reduction of drag force on the order one thousands. Hewitt [10] studied pressure gradients in liquid-liquid flows and displayed significant peaks when plotted as a function of palm oil fraction for a given velocity; the response depends on the mixing processes between the phases. Sotgia et al. [11] experimented in oil-palm oil viscosity ratios from about 560 to about 1300 and reported that the through mixing of the two liquids , which served to eliminate entrance effects on the test section, was attained in a calming section (L/D=200) before entering the test section. The structure of two-phase flow in ducts with sudden contractions and its effects on the pressure drop was studied by Guillemin et al [12].

K.D.P. Nigam [13] gave a CFD modeling of flow profiles and interfacial phenomena in two-phase flow in pipes and Variables studied include: gas velocity, volume fraction of liquid and interfacial roughness. The gas velocity was varied from 1.2 to 12.5 m/s. The liquid velocity was taken as 0.0066 and 0.1 m/s. Dongying Qian, Adeniyi Lawal [14] gave a numerical study on gas and liquid slugs for Taylor flow in a T-junction micro channel and A T-junction empty micro channel with varying cross sectional width (0.25, 0.5, 0.75, 1, 2 and 3 mm) served as the model micro-reactor. Taha Taha, Z.F. Cui [15] gave a CFD modeling of slug flow in vertical tubes and to investigate the motion of single Taylor bubbles in vertical tubes. Vimal Kumar K.D.P. Nigam [16] gave a Pressure drop and heat transfer study in tube- helical heat exchanger and the simulations were carried out in counter current mode operation with hot fluid in the tube side and cold fluid in the annulus area.

Literature scanned has not reported two-phase flow through control valves except Rani Hemamalini et. al. [17]. Hence, this study will be relevant for guiding the selection, design and setting the parameters from operating characteristics. In the present study, simulated (FLUENT6.2.16) pressure drop data's compared with experimental data's for Air –Palm oil through pipe and valve in series.

2. Experimental Setup and Procedure

A systematic diagram of the experimental setup is shown in Fig. 1. The test section is a GI-40 schedule pipe of 1meter length and 23.5 mm inside diameter, the upstream section this pipe of length 0.5 m ensures fully developed conditions. The control valve is fitted at the downstream end of the test pipe the fluids are discharged to a tank where discharge pressure is constant. The liquid is metered through Krone Marshall magnetic flow meter. Purified drier from an Ingersoll Rand compressor with a pressure regulator (0-2.5 Kg/cm^2). Pressure regulator was meter through a non return valve using a Placket Rota meter.

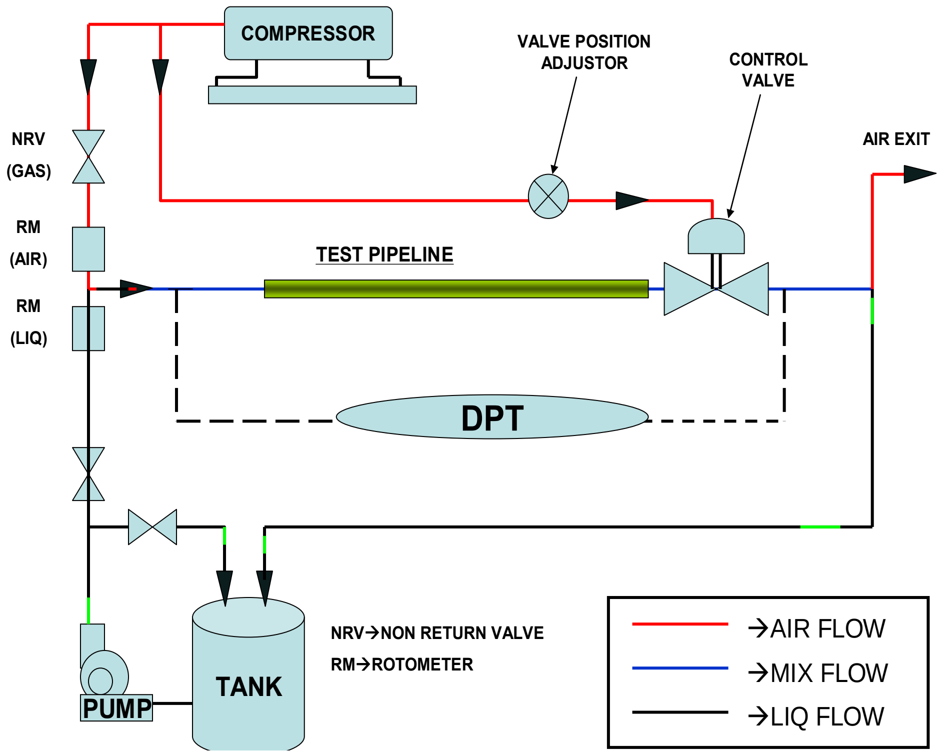


Fig. 1. Schematic Diagram of Experimental set up.

The pressure drop across the valve and the pipe was measured with a Honey Well differential pressure transducer .An electro pneumatic converter is used to actuate the pressure valve .The density and viscosity of palm oil used in the experimental are 888.25 kg/m^3& 0.044400 Ns/m^2. During experimentation the temperature of two-phase flow varies at ± 3 ^{0}C.

The experiments were carried out for four different valve openings and different volume fractions. Air and palm oil flow rates varied from 25 to 100 l/h and 50 to 150 l/h respectively (non-uniform flow) with varying valve openings from 25 % to 100 %.

The system was initially tested with palm oil (single phase flow) for different control valve openings. In subsequent experiments, the volume fraction of palm oil was varied by dispersing filtered dry air into palm oil into calming section. The air flow rates are maintained at constant pressure and measured using calibrated Rota meter.

3. Methods of Modeling

There are two well-known methods for numerically solving the set of governing equations, the finite volume and the finite element approaches. The commercial CFD software package, FLUENT 6.2.16, which is based on the finite volume approach, was used for solving the set of governing equations. Fluent provides the flexibility in choosing discretization schemes for each governing equation. The discretized equations, along with the initial and boundary conditions, were solved using the segregated solution method to obtain a numerical solution. Using the segregated solver, the conservation of mass and momentum were solved iteratively and a pressure-correction equation was used to ensure the conservation of momentum and conservation of mass. The k–ε Model was used to treat turbulence phenomena in both phases.

3.1. Governing Equations

All of CFD is based on the fundamental governing equations of fluid dynamics -the continuity, momentum and energy equation. These equations speak physics. For flow analysis energy equation is not need. They are the mathematical statements of three fundamental physical principles on which fluid dynamics is based:

The conservation equation of mass is:
The differential equation representing the mass conservation principle is

$$(\partial \rho / \partial t) + \nabla . (\rho u) = 0 \tag{3.1}$$

The differential equation governing the conservation of momentum in the x direction can be given as

$$(\partial (\rho u) / \partial t) + \nabla . (\rho u u) = - \nabla . T - (\partial P / \partial X) + B + V, \tag{3.2}$$

where B and V, are body forces and viscous forces per unit volume.

The Momentum equation shows that it depends on the density (ρ) of the fluid and viscous force (V) of the fluid.

3.2. Basic Modeling Approach

There is a variety of approaches to simulate such flow problems numerically. In this section, only a general overview of the approaches used will be given. The three most general modeling approaches

a) The volume of fluid approach (Eulerian framework for both the phase with reformulation of interface forces on volumetric basis)

b) The Eulerian-Lagrangian approach (Eulerian framework for the continuous phase and Lagrangian frame work for the dispersed phase).

c) The Eulerian-Eulerian approach (Eulerian framework for both the phases).

The volume of fluid (VOF) approach is conceptually the simplest. In this approach, the motion of all the phases is modeled by formulating local, instantaneous conservation equations for mass, momentum and energy. And generally used when it is essential to resolve small-scale fluid dynamics around individual bubbles. With VOF, it is possible to resolve small-scale vortices behind bubbles, bubble-bubble interactions (coalescence/breakup) and mass and heat transfer between bubbles and surrounding liquid. These methods can, therefore, be used to predict mass transfer coefficients and other inter phase exchange terms. However the application of VOF is usually restricted to simulations of a few bubbles due to the huge computational requirements. VOF based models can be very useful as learning tools and provide valuable information to develop appropriate closure models for Eulerian-Lagrangian and Eulerian-Eulerian approaches.

In the Eulerian-Lagrangian approach, explicit motion of the interface is not modeled. This means small-scale fluid motions around individual dispersed phase particles are not considered. Their influence is modeled indirectly while considering the motion of the dispersed phase particles. In this approach, particle-level processes such as reactions, heat and mass transfer etc. can be simulated in adequate detail. In the case of turbulent flows, it is necessary to simulate a very large number of particle trajectories to obtain meaningful averages. Therefore, even with this approach, when the number of particles to be simulated increases, computational resources becomes stretched. The approach is suitable for simulating dispersed multiphase flows containing a low (10%) volume fraction of the dispersed phases. For denser dispersed phase flows, it may be necessary to use an Eulerian-Eulerian approach.

The Eulerian-Eulerian approach models the flow of all phases in an Eulerian framework based on the interpenetrating continuum assumption. In this approach, trajectory simulation and averaging are not carried out at a computational level but are implicitly achieved at a conceptual level. This approach is the most difficult one to understand conceptually, requiring extensive modeling efforts. Various averaging issues will have to be addressed while formulating the governing equations in this approach. This approach can be applied to multiphase flow if modeled successfully.

In both the Eulerian-Lagrangian and the Eulerian-Eulerian approaches, the exchange of momentum through the interface needs to be modeled. This exchange can consist of several forces, like drag, lift, virtual mass, lubrication and wall forces. Depending on the physical problem (i.e. flow regime, gas holdup, etc.) these forces play a more or less important role. In general, it can be said that the drag force is the most important one and that this force can not be neglected. A more detailed discussion of the interface forces will be given in the next section.

3.3. Turbulence Model

Turbulence models are used to predict the effects of turbulence in fluid flow without resolving all scales of the smallest turbulent fluctuations. They have been specifically developed to account for the effects of turbulence without recourse to a prohibitively fine mesh and Direct Numerical Simulation (DNS). Most turbulence models (except Large Eddy Simulations) based on the Reynolds averaged Navier stroke's (RANS) equations are known as Statistical Turbulence Models due to the statistical averaging procedure employed to obtain the equations.

Two-equation turbulence models are very widely used, as they offer a good compromise between numerical effort and computational accuracy. The k-ε and k-ώ two equation models use the gradient diffusion hypothesis to relate the Reynolds stresses to the mean velocity gradients and the turbulent viscosity. The turbulent viscosity is modeled as the product of a turbulent velocity and turbulent length scale.

In two-equation models the turbulence velocity scale is computed from the turbulent kinetic energy. The turbulent length scale is estimated from two properties of the turbulence field, usually the turbulent kinetic energy and its dissipation rate, which are provided from the solution of its transport equation.

3.3.1. The k-ε Model

The k-ε is the most widely used model and has achieved notable success in calculating a wide variety of thin shear layer and recirculation of flows. The model performs well in confined flows where Reynolds stresses are more important. There are many versions of k- ε model available like standard k-ε model, low Reynolds number k-ε model. k is the turbulence kinetic energy and is defined as the variance of the fluctuations in velocity. It has dimensions of (L^2 T^{-2}) .ε is the turbulence eddy dissipation (the rate at which the velocity fluctuations dissipate) and has dimensions of k per unit time (L^2 T^{-3}).

The k-ε model introduces two new variables into the system of equations. The continuity equation is then:

$$(\partial \rho / \partial t) + \nabla . (\rho u) = 0 \tag{3.3}$$

The momentum equation becomes:

$$(\partial (\rho u) / \partial t) + \nabla . (\rho u u) - \nabla . (\mu_{eff} u) = \nabla p' - \nabla .(\mu_{eff} \nabla u)^T + B, \tag{3.4}$$

where B is the sum of body forces, μ_{eff} is the effective viscosity accounting for turbulence, and is the modified pressure given by p'.

$$P' = P + (2/3) \rho k \tag{3.5}$$

The k-ε model is based on the eddy viscosity concept, so that

$$\mu_{eff} = \mu + \mu_t, \tag{3.6}$$

where μ_t is the turbulence viscosity.

The k-ε model assumes that the turbulence viscosity is linked to the turbulence kinetic energy and dissipation via the relation

$$\mu_t = C_\mu \rho (k^2 / \varepsilon), \tag{3.7}$$

where Cμ is the k-ε model constant and its value is 0.09.

The values of k and ε come directly from the differential transport equations for the turbulence kinetic energy and turbulence dissipation rate:

$$(\partial (\rho k) / \partial t) + \nabla . (\rho u k) = \nabla . [\mu + \mu_t / \sigma_k) \nabla k] + P_k - \rho \varepsilon \tag{3.8}$$

$$(\partial (\rho \varepsilon) / \partial t) + \nabla . (\rho u \varepsilon) = \nabla . [\mu + \mu_t / \sigma_{\varepsilon}) \nabla \varepsilon] + (\varepsilon / k)(C_{\varepsilon 1} P_k - C_{\varepsilon 2} \rho \varepsilon), \tag{3.9}$$

where $C_{\varepsilon 1}$, $C_{\varepsilon 2}$, σ_k, and σ_ε are constants and their values are

$$C_{\varepsilon 1}=1.44;\ C_{\varepsilon 2}=1.92;\ \sigma_k=1.0;\ \sigma_\varepsilon=1.$$

4. Analysis

4.1. Model Geometry

In the simulations four different kind of grids generated for different valve opening such as 100 %, 75 %, 50 %, 25 %. The diameter of the pipe line was 2.35 cm and length of the testing section was 1 meter. The table 1 indicates the specification of grid, the grid generation for 100% valve opening is shown in Fig. 2.

Table 1. Grid Specifications.

Grid type	Maximum face area e-5 (m^2)	Cells	Faces	Nodes
25 % Valve Opening	2.88719	50427	107914	12317
50 % Valve Opening	2.793812	49776	106154	11973
75 % Valve Opening	2.901243	49877	106799	12214
100 % Valve Opening	2.665428	49362	105382	12214

4.2. Numerical Approach

The governing equations were solved with a control volume finite element method. This entails the discretization of the solution domain into a finite number of four-sided cells whose faces coincide with coordinate lines. Values of all computed variables were stored at cell centers (called nodes). The interface between the phases is arranged here to be aligned with the boundary between two rows of cells. Since the location of the interface was not known a priori, the mesh distribution over the cross-section can only be an outcome of the solution itself. What may be fixed in advance, however, was the number of mesh cells covering each phase. Under-relaxation was found to be necessary to promote the stability of the overall solution process.

The model geometry was meshed with the preprocessor GAMBIT software, then imported into processor FLUENT for calculation. The simulation results were either analyzed by FLUENT integrated postprocessor or exported into data files for other postprocessor packages. A segregated time dependent unsteady solver in FLUENT was used. The boundary conditions were: Flow rate inlet for the air and palm oil feeds and continuum specify the fluid and pressure outlet condition.

5. Results and Discussion

5.1. Pressure Drop Prediction under Different Valve Opening

Measurement of the pipe and control valve pressure drop was carried out for Palm oil flow rate ranging from 50 l/h to 150 l/h and Airflow rate ranging from 25 l/h to 100 l/h with different valve opening ranging from 25 % to 100 %. The comparison between the experiment and CFD prediction is shown Figs. 4.1 to 4.4.

Result shows that good agreement of the CFD numerical calculation when compared with experimental data. It shows that the CFD prediction by using the fluent can be used for pressure drop evaluation in pipe and control valve in series. In the CFD numerical calculations a very small pressure drop deviation were observed with less than 3% of deviation at different inlet flow rate which probably in the same magnitude of the experimental error. The k-ε Model is much less on computational time required compared to the complicated RSM turbulence model. The pressure drop increase significantly with raising the Air and Palm oil flow rate. This effect is mainly due to flow acceleration gravity and wall friction. The pressure drop decrease significantly with raising the valve opening.

Fig. 3 shows the simulated value for the effect of pressure drop for 100 % valve opening with air flow rate of 75 lph and palm oil flow rate of 100 lph.

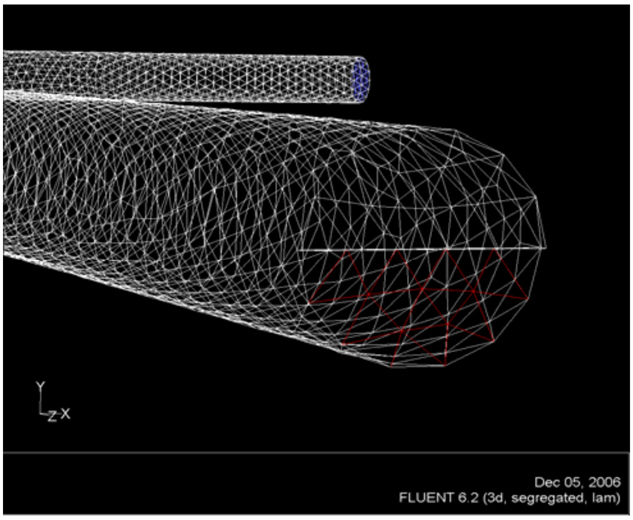


Fig. 2. Grid (100 % Valve Opening).

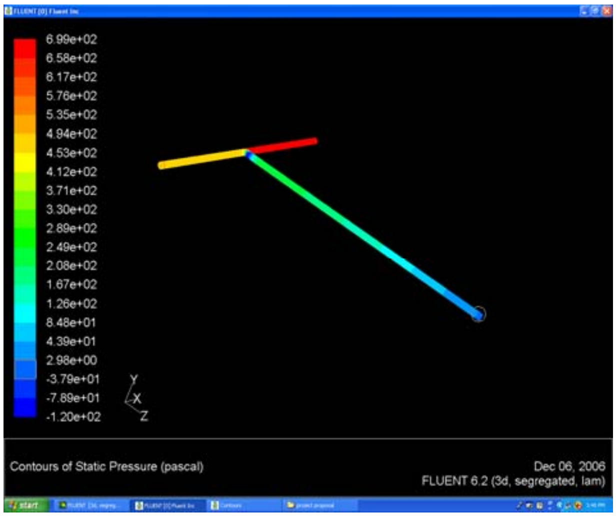


Fig. 3. Effect of pressure drop for Valve Opening100% (Air flow rate 75 l/h, Palm oil 100 l/h).

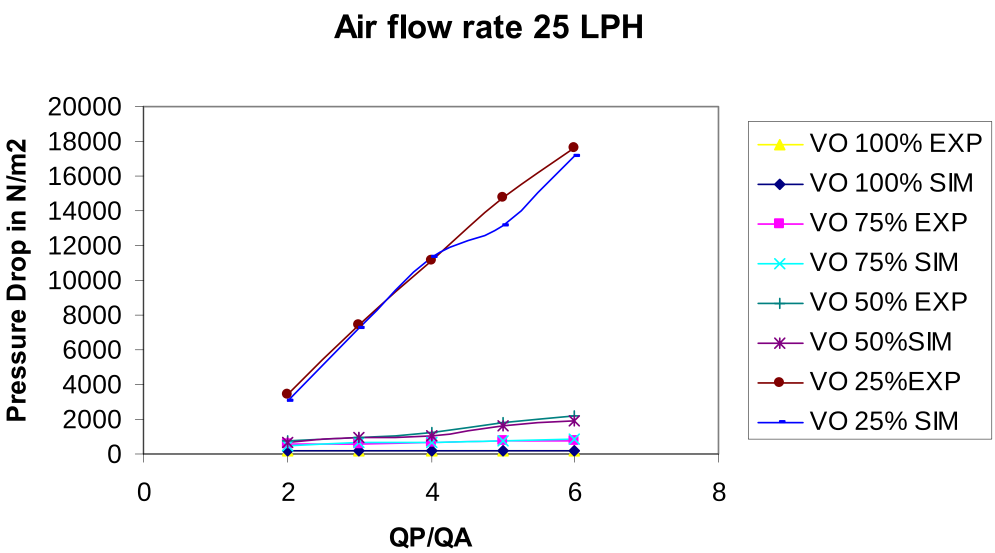


Fig. 4.1. Effect of Total Pressure Drop for Air-Palm oil for different valve opening at Air flow rate 25LPH.

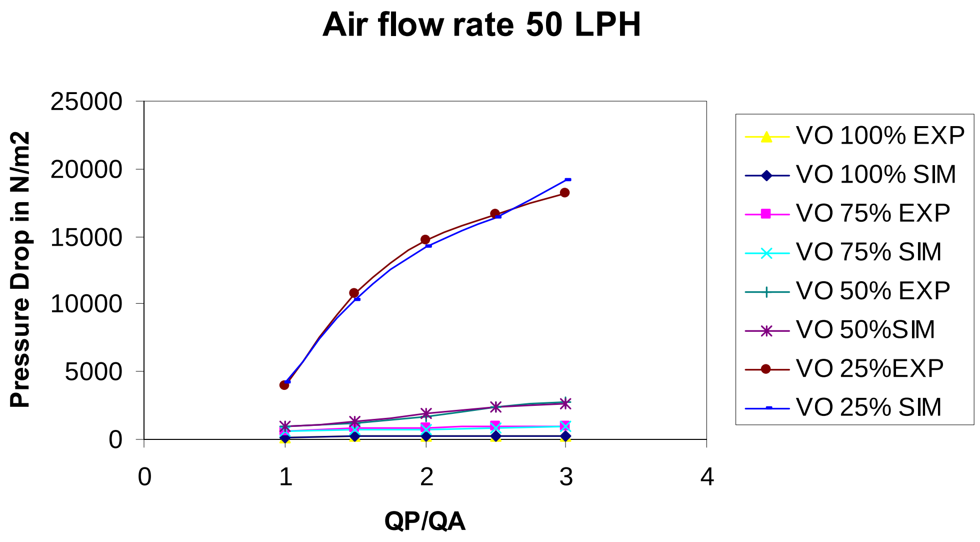


Fig. 4.2. Effect of Total Pressure Drop for Air-Palm oil for different valve opening at Air flow rate 50LPH.

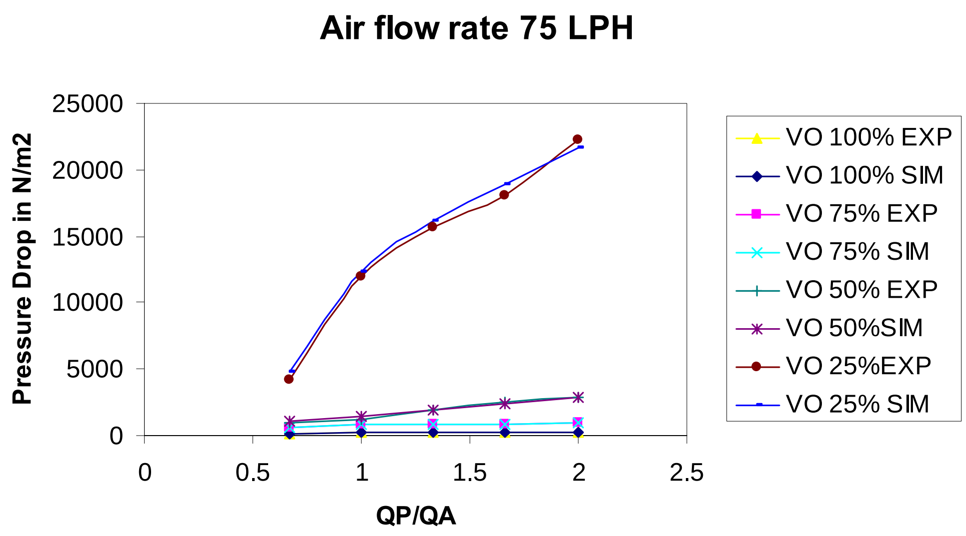


Fig. 4.3. Effect of Total Pressure Drop for Air-Palm oil for different valve opening at Air flow rate 75LPH.

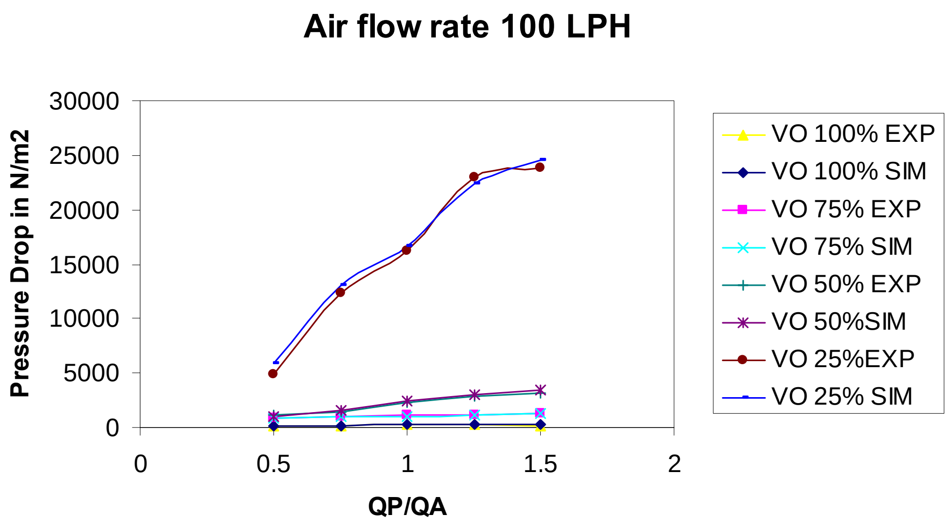


Fig. 4.4. Effect of Total Pressure Drop for Air-Palm oil for different valve opening at Air flow rate 100LPH.

6. Conclusion

Experimental and Modeling of Air–Palm oil flow through pipe and valve in series has been analyzed in this study. Computational fluid dynamics software (FLUENT 6.2.16) was used for numerical computation. The numerical profile of the pressure drop in pipe and control valve in series has been compared to experimental profiles. Extensive calculations were carried out for the cases of various Air and Palm oil flow rates and Valve position. The agreement between experiment and simulation was found to be satisfactory. The simulations in the Air- Palm oil system validate the concept of pressure drop for present studies. The agreement of the predicted data and experimental data are quite reasonable.

Although present formulation is rather complex and demands much computational time, due to the mixed nature of the multi phase model and the fine grids required for its implementation, it does appear to demonstrate that the CFD technique can be successfully applied to two phase (Air –Palm oil) flow in pipe and control valve in series.

Nomenclature

u Velocity component in (m/s)
P Pressure in (pa)
B Body forces per unit volume
V Viscous forces per unit volume
P' Modified pressure (pa)
QA Air flow rate (l/h)
QP Palm oil flow rate (l/h)

Greek Symbols

ρ Density of fluid (kg/m3)
k Turbulence Kinetic energy ($J\ kg^{-1}$)
ε Turbulence kinetic Dissipation ($J\ kg^{-1}\ s^{-1}$)
μ_{eff} Effective viscosity (Ns/m^2)
μ_t Turbulence viscosity (Ns/m^2)

References

[1]. Baker, O., Design of pipe line for the simultaneous flow of oil and gas, *Oil Gas Journal,* 53, 1954, pp. 184-185.
[2]. Hoogendoorn, G. L., Gas-liquid flow in horizontal pipes, *International Journal of Chemical Engineering Science,* 9, 1959, pp. 205- 217.
[3]. Vermeulen, L. R., Ryan, J. T., Two-phase slug flow in horizontal and inclined tubes, *Canad. Journal of Chemical Engineering*, 49, 1971, pp. 195-201.
[4]. Beretta, A., Ferrari, P., Galbaatti, L., A. Andreni, P., Horizontal oil-palm oil flow in small diameter tubes; pressure drop, *Int. Commun. Heat Mass Transfer,* 24, 1997, pp. 231-239.
[5]. Awwad, A., Xin, R. C., Dong, Z. F., Ebadian, M. A., Soliman, H. M., Flow patterns and Pressure drop in air/palm oil two-phase flow in horizontal helicoidal pipes, *Trans. ASME*, *J. Fluids Eng*., 117, 1995, pp. 720-726.
[6]. Kordyban, E., Horizontal slug flow: a comparison of existing theories, *Trans. ASME, J. Fluids Eng*., 112, 1990, pp. 75-83.
[7]. Chisholm, D., Pressure gradient due to friction during the flow of evaporating two-phase mixtures in smooth tubes and channels, *J. Heat Mass Transfer,* 16, 1972, pp. 347-356.
[8]. Oliemans, R. V. A., Ooms, G., Core –annular flow of oil and palm oil through pipelines, *Multiphase Sci. Technol*., 2, 1986, pp. 427-477.
[9]. Bai, R., Chen, K., Joseph. D, Lubricated pipelining; stability of core-annular flow Part 5: Experiments and comparison with theory, *J. Fluid Mech*., 240, 1992, pp. 97-142.
[10].Hewitt, G. F., From gas-liquid to liquid-liquid flow: a difficult journey? In: *Proceedings of International Symposium on Liquid-Liquid Two-Phase Flow and Transport Phenomena*, Antalya, Turkey, 1997, pp. 3-19.

[11].Sotgia, G., Sparta, G., Vensola, E., Tartarini, P., Experimental results on Pressure reductions and flow regime transitions in oil-palm oil mixtures, in *Proceedings of the 5th World Conference on Experimental Heat Transfer, Fluid Mechanics and Thermodynamics*, Thessaloniki, Greece, 2001, pp. 1763-1770.
[12].Guglielmini, G., Muzzio, A., Sotgia G., The structure of two-phase flow in ducts with sudden contractions and its effects on the pressure drop, *Invited Lecture to Experimental Heat Transfer, Fluid Mechanics and Thermodynamics,* ETS ed. 1997.
[13].Subhashini Ghorai, Nigam, K. D. P., CFD modeling of flow profiles and Interfacial phenomena in two-phase flow in pipes, *Journal of Chemical Engineering and Processing,* 45, 2005, pp. 55-65.
[14].Dongying Qian, Adeniyi Lawal, Numerical study on gas and liquid slugs for Taylor flow in a T-junction micro channel, *Journal of Chemical Engineering Science,* 61, 2005, pp. 7609-7625.
[15].Taha Taha, Cui , Z. F., CFD modeling of slug flow in vertical tubes, *Journal of Chemical Engineering Science*, 61, 2006, pp. 676-687.
[16].Vimal Kumar, Supreet Saini, Manish Sharma, Nigam K. D. P., Pressure drop and heat transfer study in tube-in-tube helical heat exchanger, *Journal of Chemical Engineering Science,* 61, 2005, pp. 4403-4416.
[17].Rani Hemamalini, R., Partheeban, P., Sarat Chandrababu, J., Sundaram S., The effect on pressure drop across horizontal pipe and control valve for air/palm oil two- phase flow, *International Journal of Heat and Mass Transfer*, 48, 2004, pp. 2911-2921.
[18].Lockhart, R. J., and R. C. Martinelli, Proposed Correlation of Data for Isothermal Two-phase, Two-Component Flow in Pipes, *Chemical Engineering Progress*, 45, 1, 1949, pp. 39-48.

___

Guide for Contributors

Aims and Scope

Sensors & Transducers Journal (ISSN 1726-5479) provides an advanced forum for the science and technology of physical, chemical sensors and biosensors. It publishes state-of-the-art reviews, regular research and application specific papers, short notes, letters to Editor and sensors related books reviews as well as academic, practical and commercial information of interest to its readership. Because it is an open access, peer review international journal, papers rapidly published in *Sensors & Transducers Journal* will receive a very high publicity. The journal is published monthly as twelve issues per annual by International Frequency Association (IFSA). In additional, some special sponsored and conference issues published annually.

Topics Covered

Contributions are invited on all aspects of research, development and application of the science and technology of sensors, transducers and sensor instrumentations. Topics include, but are not restricted to:

- Physical, chemical and biosensors;
- Digital, frequency, period, duty-cycle, time interval, PWM, pulse number output sensors and transducers;
- Theory, principles, effects, design, standardization and modeling;
- Smart sensors and systems;
- Sensor instrumentation;
- Virtual instruments;
- Sensors interfaces, buses and networks;
- Signal processing;
- Frequency (period, duty-cycle)-to-digital converters, ADC;
- Technologies and materials;
- Nanosensors;
- Microsystems;
- Applications.

Submission of papers

Articles should be written in English. Authors are invited to submit by e-mail editor@sensorsportal.com 6-14 pages article (including abstract, illustrations (color or grayscale), photos and references) in both: MS Word (doc) and Acrobat (pdf) formats. Detailed preparation instructions, paper example and template of manuscript are available from the journal's webpage: http://www.sensorsportal.com/HTML/DIGEST/Submition.htm Authors must follow the instructions strictly when submitting their manuscripts.

Advertising Information

Advertising orders and enquires may be sent to sales@sensorsportal.com Please download also our media kit: http://www.sensorsportal.com/DOWNLOADS/Media_Kit_2008.pdf