VOL. 11, NO. 20, OCTOBER 2016 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences

www.arpnjournals.com

CFD SIMULATION OF BUBBLY DIESEL-CNG FLOW IN A HORIZANTAL PIPE UNDER THE INFLUENCE OF A MAGNETIC FIELD

Hasanain A. Abdul-Wahhab, A. Rashid. A. Aziz, Hussain H. Al-Kayiem and Mohammad S. Nasif
Mechanical Engineering Department, Universiti Teknologi Petronas, Seri Iskandar, Perak, Malaysia
E-Mail: rashid@petronas.com.my

ABSTRACT

The pre-mixing Diesel-CNG was suggested to increase the air quantity inside the cylinder more than if a mixture of CNG-air is injected in the engine which may result is a rich mixture, directly. The objective of the present paper is to study the behaviour of gas bubbles in the liquid fuels and the ways to control the bubbles sizes prior to injection by an external magnetic field. Numerical simulation of Diesel-CNG bubbly flow with effecting magnetic field is presented in this paper. The three-dimensional incompressible Navier-Stokes equations have been used to solve the CNG-Diesel two phase flows in 20 mm ID horizontal pipe. The simulation was carried out using ANSYS fluent software and the flow field discretization was achieved by Volume-of-fluid method (VOF) technique. The numerical results showed that CNG bubbles tend to move toward the upper wall under the buoyancy force effect and these bubbles grow to a bigger size and expand vertically in the Diesel flow before it breaks away with effecting magnetic field 0.4 to 0.8 Tesla, respectively, and the gas volume fraction values increased with noted to by increasing the magnetic intensity. The laminar behaviour of the flow changed in upper zone from the pipe to increasing gas volume fraction, while the axial Diesel velocity decreases and the profiles tend to flatten with increasing of the magnetic field strength. The numerical results have been validated by comparing the computational results with experimental results reported in the literature where a good agreement was achieved.

Keywords: two-phase flow, bubbly flow, magnetic field, volume-of-fluid.

INTRODUCTION

The analysis of liquid-gas flow dynamics under electric and magnetic fields has become an important effect in the design and control of many electric-apparatus systems, such as magneto hydrodynamic problems (MHD), gas and oil facilities, processing nuclear reactors, and MHD power stations Ki H. [1]. In Diesel engines, one of several solutions of improving performance and reducing emissions that have been implemented in recent was blending gas fuel (as CNG), the pre-mixing Diesel and CNG was suggested to increase the air quantity inside the cylinder more than if a mixture of CNG-air is injected in the engine which may result is a rich mixture, this mixture requires a study of the behaviour of gas bubbles in the liquid fuels and ways to control on their size before arriving in the injection system through the influence of external forces, such as a magnetic field.

The horizontal bubbly flow model is described by the appearance of bubbles sparse in a continuous liquid phase, with their maximum volume being much smaller than the diameter of the containing pipe; this particular flow has met less attention when compared to vertical bubbly flow. On the other hand, complexity in this case; the motion of bubbles towards the upper side of the horizontal pipe, due to the buoyancy effect, which consequently causes a highly non-symmetric volume fraction distribution in the pipe cross-section Ekambara K., *et al*. [2]. Ishimoto *et al.* [3] have studied experimentally the effect of non-uniform magnetic field on a bubbly flow; and compared his results with the numerical and experimental results reported by Hnat and Buckmaster [4], where excellent agreement has been achieved. Bhaga and Weber [5] was studied the shape and velocities of bubbles in viscous liquids by experiment, while Ryskin and Leal [6] improve a numerical method to compute the motion of a bubble rising in the liquid. Most popular numerical methods for interface tracking are Volume-of-Fluid (VOF) technique implemented in ANSYS Fluent [2] or Level-Set (L-S) method in COMSOL Multiphysic [7].

The literature review showed that various researches have studied the effects of uniform or non-uniform magnetic field on vertical two-phase flows. But effects of change intensity of uniform magnetic field on bubbly flow in horizontal pipe, has not been studied in details. Hence in this study, effect of change intensity uniform magnetic field on bubbly flow (Diesel-CNG) in horizontal pipe was studied. Bubbly flow behavior of CNG in Diesel is to cover a wide range of two phase flow properties (velocities of liquid and gas, volume fraction, and liquid-gas properties). For that, bubbles of CNG in Diesel flow have been simulated and this paper presents the simulation results at various magnetic field intensity influencing on Diesel-CNG flow. The simulation was carried out using that ANSYS fluent software, Volume-of-fluid method.

COMPUTATIONAL MODELLING

The computational fluid dynamics (CFD) code includes the numerical solutions of the fundamental laws to fluid dynamics namely the continuity, momentum, energy, species, and turbulent equations [8]. The FLUENT software package was used to accomplish this job.

Governing equations

Bubble flow dynamics in electrically conductive liquid in external DC electromagnetic field can be characterized by the following set equation:

$$\nabla^2 \vec{B}/(\mu_o \sigma_l) + (\vec{B}.\nabla)\vec{U} - (\vec{U}.\nabla)\vec{B} = 0 \quad (1)$$

$$\vec{J} = \sigma_l(-\nabla\varphi + [\vec{U} \times \vec{B}]) \quad (2)$$

Where $\vec{B}$ is magnetic field induction, μ_o - magnetic constant, σ_l - electrical conductivity of liquid, $\vec{U}$ - velocity, the pair of equations describes electromagnetic nature of the process: Amperes circuital law Equation (1) (where displacement currents are neglected), and Ohm's law Equation (2).

$$\nabla.\vec{U} = 0 \quad (3)$$

$$\frac{\partial \vec{U}}{\partial t} + (\vec{U}.\nabla)\vec{U} = -\frac{1}{\rho_l}\nabla p + \upsilon_l \nabla^2 \vec{U} + \vec{K} \quad (4)$$

Where p is Pressure, υ_l is the kinematic viscosity of the liquid, $\vec{K}$ is the sum of volume forces, which are:

$\vec{K}_{fric} = N(\Delta\vec{U}_{lg})/\rho_l$ is the friction force,

$\vec{K}_{Lor} = (\vec{J} \times \vec{B})/\rho_l$ is the Lorentz force,

$\vec{J}$ is the current density, and

$\vec{K}_g$ is the drag force. Equation 3 is denoting the hydrodynamic processes of conductive viscous liquid, and Equation 4 is denoting the momentum balance.

Volume of fluid method

The phase flow distribution can be characterized by a phase indicator function V_F (r_i, t) (i^{th} mesh element is given a scalar value V_F). The indicator function has value one or zero when a control volume is entirely filled with one of the phases and a value between one and zero if an interface is present in the control volume. Hence, the phase indicator function has the properties of volume fraction. In general, interface dynamics is characterized by the following transport equation:

$$\frac{\partial V_F}{\partial t} + \vec{U}.\nabla(V_F) = 0 \quad (5)$$

METHODOLOGY

The methodology adopted for the present work is as follows. Diesel flow through horizontal pipeline (20 mm diameter and 100 mm length) and CNG gas injected axially (1.0 mm nozzle diameter). An external magnetic field affects the flow perpendicular to the two phase flow line to study the CNG bubbles behavior during work conditions, Figure 1 show the suggested scheme, which includes the following steps:

- Solid modelling of a horizontal pipeline with axial gas injector
- Mesh generation.
- Solution of the governing equations with appropriate boundary conditions.
- Comparison of the simulated results with the available experimental results reported in the literature.

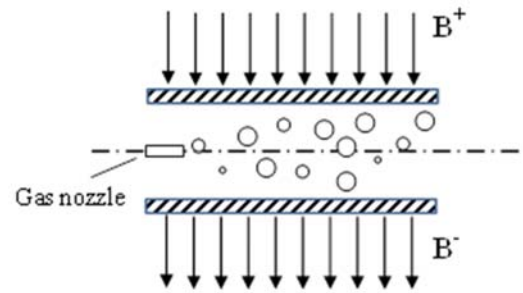


Figure-1. Show the suggested scheme.

The study is expected to explore the potential of using ANSYS Fluent software tools for analysis of two phase flow characteristics. The package includes user interfaces to input problem parameters and to examine the results. The three-dimensional model of the mixer is developed by using the pre-processor CFD software. The number of cells used in this model was 297,423. Mesh refinement investigation has been carried out to optimize the number of cells used. It has been found that increasing the number of cells to 317,688 and 347,730 has no effect on the results accuracy. Hence, 297, 423 were selected to be the optimum number of cells that can be used in the simulation. Figure-2 shows (a) the solid modelling and (b) the mesh for case under study.

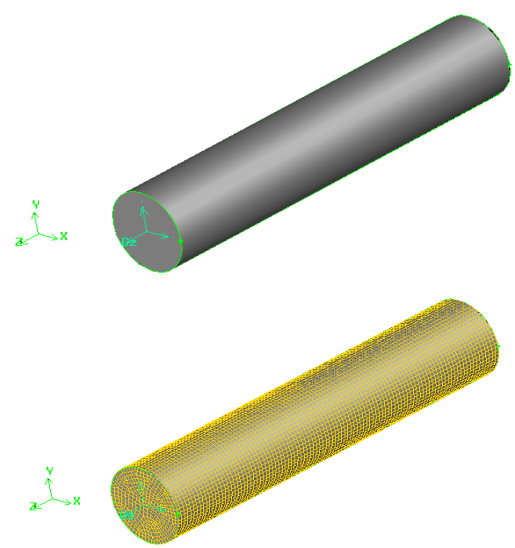


Figure-2. Shows (a) the solid modelling, and (b) the normal type mesh for case under study.

Boundary conditions

Consider unsteady, laminar, hydro-magnetic, fully developed, CNG gas injection in Diesel flow in a horizontal pipeline. A uniform transverse magnetic field is

applied normal to the flow direction (see Figure-1). The Diesel phase is assumed to be electrically conducting depending on the sulfur content of the fuel. Assuming no electric field with exist and the hall impact of MHDs is negligible. The governing equations are based on the conservation laws of mass and momentum of both phases. Attach boundaries are specified on the coincident cell face near the cells around CNG bubbles. No slip wall boundary condition in conjunction with logarithmic law of wall is used. Table-1 property of Diesel fuel and CNG.

Table-1. Property of diesel fuel and CNG.

Parameter	Diesel	CNG
Density (kg/m^3) (23 ^{o}C)	**840**	**0.72**
Viscosity (N.s/m^2) (23 ^{o}C)	0.0024	7.8*10^{-6}
Carbon (%, w/w)	86.83	73.3
Hydrogen (%, w/w)	12.72	23.9
Oxygen (%, w/w)	1.19	0.4
Sulphur (%, w/w)	0.25	ppm < 5
Electric Conductivity(1/Ωm)	25	-
Relative permittivity	2.0-2.2	-

The inlet velocity of Diesel was 0.18 m/s and CNG injected to the flow, the CNG velocity equal zero when (t= 0) (in injection time). Magnetic field intensity changed (0, 0.4, and 0.8 Tesla) respectively.

RESULTS AND DISCUSSIONS

Sensitivity analysis on CNG volume fraction profile

In order to understand the effect of different forces, the numerical simulations have been carried out for three different cases. In the first case, the 3D simulations are carried out taking friction force and drag force into account while magnetic force (Lorentz force) neglected (*B*= 0), the predicted volume fraction profile shows a peak in the top of the pipe, where CNG bubbles tend to migrate toward the upper wall, Figure 3 shows counters of CNG volume fraction in Diesel flow at (*B*=0) and during different time: (a) 0.1 sec, (b) 0.3 sec, (c) 0.5 sec, (d) 1.0 sec, (e) 1.5 sec.

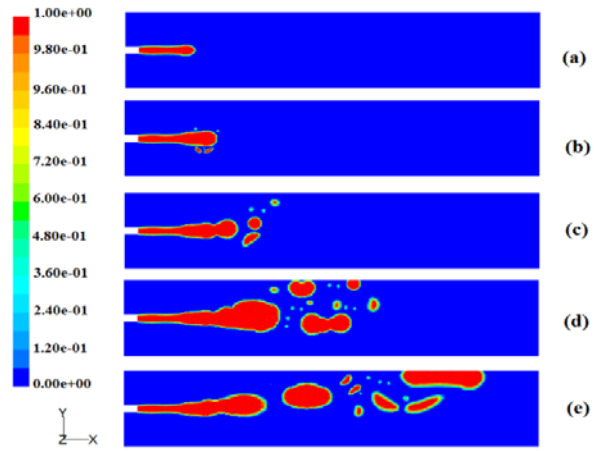


Figure-3. Shows counters of CNG volume fraction to Diesel-CNG bubbly flow at (B=0) and during different time: (a) 0.1 sec, (b) 0.3 sec, (c) 0.5 sec, (d) 1.0 sec, (e) 1.5 sec.

In second case, the simulations were carried out including magnetic force with (B= 0.4 Tesla). The result shows significant change and the behaviour was basically same tend to migrate toward the upper wall but CNG bubbles grows to a bigger volume and expands vertically in the Diesel flow before it breaks away in 0.4 Tesla, Figure-4 shows counters of CNG volume fraction in Diesel flow at (B=0.4 Tesla) and during different time: (a) 0.1 sec, (b) 0.3 sec, (c) 0.5 sec, (d) 1.0 sec, (e) 1.5 sec. In third case, when the magnetic force was increased (B=0.8 Tesla), the CNG volume fraction profile shows acceleration in this mechanism with the same behaviour for the elongation of the bubble in a vertical direction of Diesel flow. These results show good agreement for effect magnetic force with experimental and numerical data for Ishimoto *et al*. [3], Hnat and Buckmaster [4] in vertical bubbly flow, and Fern´andez D. [9].

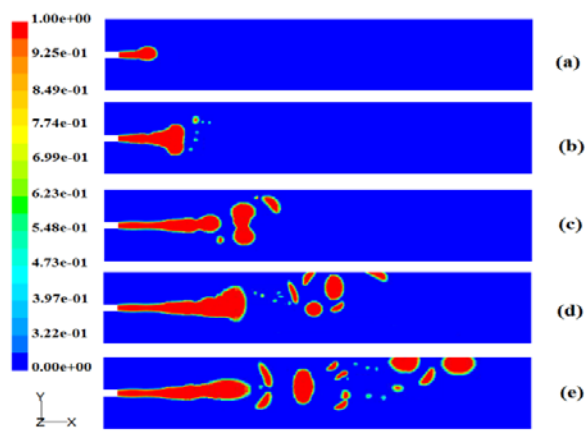
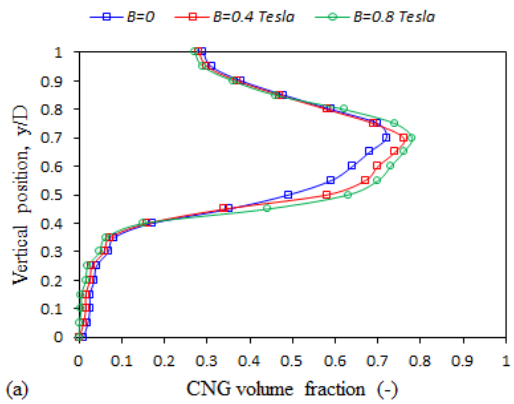
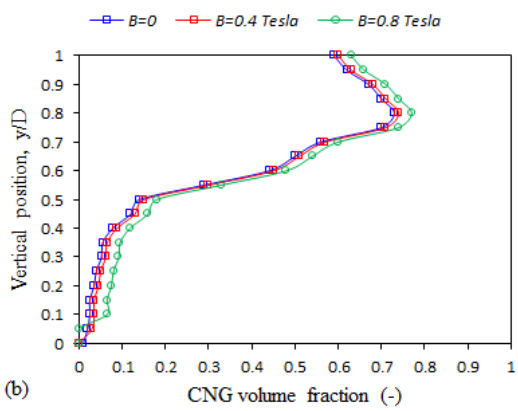


Figure-4. Shows counters of CNG volume fraction to Diesel-CNG bubbly flow at (B=0.4 Tesla) and during different time: (a) 0.1 sec, (b) 0.3 sec, (c) 0.5 sec, (d) 1.0 sec, (e) 1.5 sec.

Figure-5 shows the comparison of the predicted CNG volume fraction profiles with three different positions (a: x/L = 0.4, b: x/L = 0.65, c: x/L = 0.85) with changing magnetic field intensity (B = 0, 0.4, 0.8 Tesla). It might be seen from figures that the local CNG volume fraction profile shows a peak at the upper side of the horizontal pipe , where gas bubbles tend to migrate toward the upper wall and this behaviour increase with increasing magnetic intensity and in same time when the flow tend to the outlet edge but this changing was gradually. This peak can be attributed to the increased hydraulic resistance of the liquid path between the bubble and the wall which may cause a sharp decline in volume fraction and it increases with increasing magnetic field effect. The similar observation was made experimentally by Kocamustafagullari and Wang [10], Kocamustafaogullari and Huang [11], and Iskandrani and Kojasoy [12].





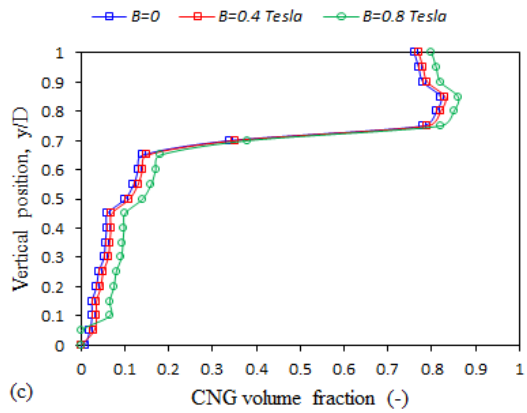


Figure-5. Shows profile of CNG volume fraction with vertical position at (B=0, 0.4, 0.8 Tesla) and during different location along pipe: (a) x/L=0.4, (b) x/L=0.65, (c) x/L=0.85.

Effect of magnetic field on diesel and CNG velocity

Figure-6 show the comparison of predicted axial Diesel velocity profiles for different values of magnetic field intensity (*B* = 0, 0.4, 0.8 Tesla) and during different location along pipe: (a) x/L = 0.4, (b) x/L = 0.65, (c) x/L = 0.85. In a single liquid phase moves in the pipe, the velocities of liquid in the pipe top region will be equal to the velocities in the bottom region. But these results show that the axial Diesel velocity profile has a slight degree of asymmetry due to the presence of CNG flow. The degree of asymmetry changes with changing the position along pipe to the outlet and in same time the axial Diesel velocity decreased with increasing magnetic intensity. The diesel speed in the upper region of the pipe may be somewhat bring down over of the lower region. This attributed to bigger volume fraction of gas in the upper region which is the reason for the asymmetric distribution of the diesel velocity and this behaviour is increasing with progress in different locations along the pipe. The slip velocity, because of the huge difference in densities between phases, is an important characteristic of two-phase flow, while the axial Diesel velocity profile tend to flatness with increasing magnetic intensity and the reason for this big slip velocity is that gas moves with less limitation by liquid and liquid velocity tends to decrease.

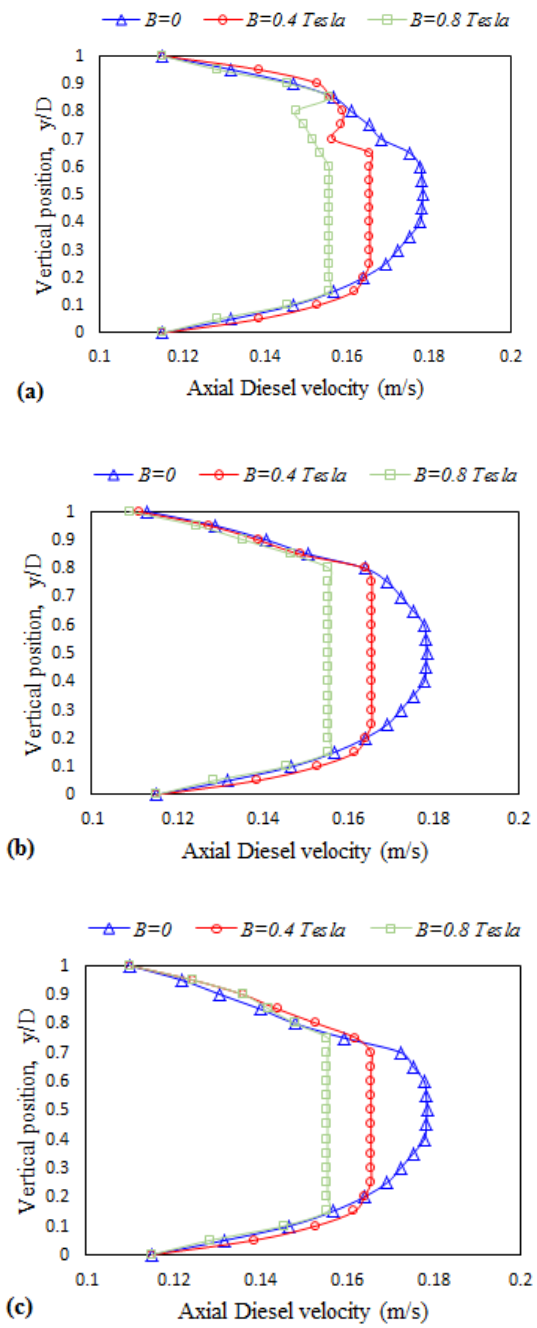


Figure-6. Shows profile of axial diesel velocity with vertical position at (B=0, 0.4, 0.8 Tesla) and during different location along pipe: (a) x/L=0.4, (b) x /L=0.65, (c) x/L=0.85.

CONCLUSIONS

The effect of change in intensity of a uniform magnetic field on bubbly flow (Diesel-CNG) in a horizontal pipe has been investigated through computational simulation. The Volume-of-fluid method and three-dimensional incompressible Navier-Stokes equations have been used for simulating the motion of CNG bubbles in Diesel, in horizontal pipe flow, under magnetic field effect. The computational results have been compared with experimental data from another work in the literature [2-4], [10-12] and a good agreement has been shown. The followings could be concluded from the analysis of results:

- CNG bubbles tend to migrate toward the upper wall under buoyancy effect and these bubbles grow to a bigger volume and expand in the Diesel flow field before they break away.
- Gas volume fraction values increased with increasing magnetic intensity.
- The laminar behaviour of the flow is changed in the upper zone of the pipe leading to increasing of the gas volume fraction.
- The axial liquid velocity decreases and the profiles tend to flatten with increasing of the magnetic field strength.

It is highly recommended to visualize the CNG injection in liquid Diesel flow in horizontal pipe under the effect of various magnetic field strengths.

ACKNOWLEDGEMENTS

The authors are obliged to the Universiti Teknologi PETRONAS for providing support through the center for automotive research and energy management (CAREM).

REFERENCES

[1] Ki H. 2010. Level set method for two-phase flows under magnetic fields. Computer Physics Communications. 999-1007.

[2] Ekambara K., Sanders R.S., Nandakumar K., and Masliyah J.H. 2008. CFD simulation of bubbly two-phase flow in horizontal pipes. Chemical Engineering Journal. 144: 277-288.

[3] Ishimoto J., Okubo M., Kamiyama S., and Higashitani M. 1995. Bubble behaviour in magnetic fluid under a nonuniform magnetic field. International Journal of JSME. 38(3): 382-387.

[4] Hnat J.G. and Buckmaster J.D. 1976. Spherical cap bubbles and skirt formation. The Physics of Fluids. 19: 182-194.

[5] Bhaga D. and Weber M.E. 1981. Bubbles in viscous liquids: shapes, wakes and velocities. Journal of fluid Mechanics. 105: 61-85.

[6] Ryskin G. and Leal L.G. 1984. Numerical solution of free boundary problems in fluid mechanics, part 2, Buoyancy-driven motion of a gas bubble through a

quiescent liquid. Journal of Fluid Mechanics. 148: 19-35.

[7] Osher S. and Sethian J. A. 1988. Fronts propagating with curvature dependent speed: algorithms based on Hamilton-Jacobi formulations. Journal of Computational Physics. 79: 12-49.

[8] Rahim A. R. 2008. Design and simulate mixing of compressed natural gas with air in a mixing device. Malaysian Technical Universities Conference on Engineering and Technology. Perlis. 2: 99-104.

[9] Fern´andez D., Martine M., Meagher A., Mobius, M.E. and Coey J.M.D. 2012. Stabilizing effect of a magnetic field on a gas bubble produced at a microelectrode. Journal Electrochemistry Communications. 1-13.

[10] Kocamustafaogullari G. and Wang Z. 1991. An experimental study on local interfacial parameters in a horizontal bubbly two-phase flow. Int. J. Multiphase Flow. 17: 553-572.

[11] Kocamustafaogullari G. and. Huang W.D. 1994. Internal structure and interfacial velocity development for bubbly two-phase flow. Nucl. Eng. Des. 151: 79-101.

[12] Iskandrani A. and Kojasoy G. 2001. Local void fraction and velocity field description in horizontal bubbly flow. Nucl. Eng. Des. 204: 117-128.