Prediction of Separation Length of Turbulent Multiphase Flow Using Radiotracer and Computational Fluid Dynamics Simulation

S.Sugiharto[1*], R. Kurniadi[2], Z. Abidin[3], Z. Stegowski[4] and L. Furman[4]

[1]*Center for Application of Isotopes and Radiation Technology, National Nuclear Energy Agency, Jl. Lebak Bulus Raya No 49, Jakarta 12440, Indonesia*

[2]*Department of Physics, Faculty of Mathematics and Natural Sciences, Bandung Institute of Technology, Jl. Ganesha 10, Bandung 40132, Indonesia*

[3]*Star Energy, Wisma Barito, Jl. Let. Jen. S. Parman Kav. 62 – 63, Jakarta 11410, Indonesia*

[4]*AGH University of Science and Technology, Faculty of Physics and Applied Computer Science, al. A. Mickiewicza 30, 30-059 Krakow, Poland*

ARTICLE INFO

Article history:
Received 04 December 2012
Revised in Revised form 29 January 2013
Accepted 09 March 2013

Keywords:
Multiphase
Radiotracer
Turbulent
CFD
Separation length
Mixture model

ABSTRACT

Multiphase flow modeling presents great challenges due to its extreme importance in various industrial and environmental applications. In the present study, prediction of separation length of multiphase flow is examined experimentally by injection of two kinds of iodine-based radiotracer solutions into a hydrocarbon transport pipeline (HCT) having an inner diameter of 24 in (60,96 m). The main components of fluids in the pipeline are water 95%, crude oil 3% and gas 2%. A radiotracing experiment was carried out at the segment of pipe which is located far from branch points with assumptions that stratified flows in such segment were achieved. Two radiation detectors located at 80 and 100 m from injection point were used to generate residence time distribution (RTD) curve resulting from injection of radiotracer solutions. Multiphase computational fluid dynamics (CFD) simulations using Eulerian-Eulerian control volume and commercial CFD package Fluent 6.2 were employed to simulate separation length of multiphase flow. The results of study shows that the flow velocity of water is higher than the flow rate of crude oil in water-dominated system despite the higher density of water than the density of the crude oil. The separation length in multiphase flow predicted by Fluent mixture model is approximately 20 m, measured from injection point. This result confirms that the placement of the first radiation detector at the distance 80 m from the injection point was correct.

INTRODUCTION

Fluid flow is commonly encountered in industrial and environmental engineering. Long distance transfer pipelines are used in petrochemical, petroleum and process industries to transport various types of fluids such as air, gasand fluids. Those fluid flow together with different velocities resulting in stratified flows. In such situation a multiphase flow occurs. As the flow of a fluid is related to its hydrodynamic properties, an understanding of flow behavior under the circumstance environment is essential for the safe design and efficient operation of the transfer pipeline.

The fundamental knowledge of multiphase flow is still not well understood, because [1]: (i) In multiphase flow many types of flow (e.g. gas-solid, gas-liquid, liquid-liquid, etc) can occur simultaneously, and within each flow type several flow regime (annular flow, jet flow, slug flow, etc) canoccur. (2) The physical laws and mathematical treatment of interface dynamics of the flow are still based on simplified assumption. Empirical data of interface interaction based on experiment are rare. (3) Numerical calculations for solving the governing equation and closure problems are extremely complex. In such situation, additional computational models for solving turbulent problems are essential.

Despite major difficulties identified, significant progresses have been made in various areas of multiphase flow. Several empirical correlations and phenomenological models have been introduced for the prediction of stratified flow. In many cases, interaction mechanisms of stratified flow are developed based on dominant properties of fluids. Shear distribution on the walls and on the

*Corresponding author.
E-mail address:sugi@batan.go.id

interfaces of flows are simplified by considering multiphase flows as single phase flows which are separated each other [2,3].

This paper is driven by the unexpected findings of visual inspection in the field. Plant engineers of a petroleum exploration company located in Riau province found that crude oil always arrived later in gathering tank than water. In other words, the speed of crude oil is always slower than speed of water in the pipeline. The exploration activities has been performed for more than fifty years, and presently to keep the pressure high enough in the oil well, a scheme called enhanced oil recovery (EOR) has been implemented by the company to lift upthe crude oil from reservoir. Water pumped down will sometimes mix with the crude oil; therefore, measured fluids in the hydrocarbon transport (HCT) pipeline from current exploration consists of approximated water 95%, crude oil 3% and gas 2%. Other substances may existin this multiphase system but their amounts are negligible [4].

This paper partly focuses on describing the measurement of residence time distribution of multiphase fluid flow in HCT pipelines using radiotracer technique. Tracers has long been used for parameters characterization of flow system [5,6]. Inprinciple, a tracer experiment is based on impulse-response method where a tracer is injected at the inlet of a system, and the concentration-time curve *C(t)* is recorded at the outlet [7]. Among various available tracers, γ–emitting radioisotopes offer several advantages over conventional ones such as a high detection sensitivity, in-situ detection, availability of radioisotopes which are compatible with the traced materials in the system, and stability under hostile industrial environments. Nowadays, necessary tools for system measurement and system analysis by radiotracer techniques are available [8-10].

It has been observed that the outcome of the tracer experiment is a response curve that does not give insight into the flow field. However, additional fundamental informations about flow process can be obtained through numerical simulation tools such as computational fluid dynamics (CFD). An increasing number of studies on flows in reactors using CFD have been published in the last decade [11-13]. The detailed information provided on flow field is required, for instance, to resolve chemical phenomena. CFD softwares uses physical laws for describing the flow mechanics in a system. Turbulent models have to be supplemented to the software when the flow in investigated system is turbulent. Among the models used for modeling turbulence, the standard $k-\varepsilon$ model is the most established, mainly due to its robustness and for its good convergence even for complex turbulent flows. The application of CFD RANS (Reynolds-averaged Navier-Stokes) equation based model to turbulent flow with mass transfer requires definition of empirical constants and functions in the turbulent transport equations.

Flow velocities of each fluid phase in HCT pipeline using radiotracer method has been studied previously [14]. Residence time distribution (RTD) simulation based on tanks-in-series model was implemented for calculating the model parameter to quantify the mixing level of each fluid flow in the system. In this work, the RTD data obtained from the aforementioned previous study is further elaborated using CFD simulation in order to obtain information on separation length of multiphase flows. To our understanding, this approach is novel to this work. One likely benefit of this study is a deeper insight into the profile of flow structure of each fluid in multiphase flow.

The objective of the current study is twofold: (1) To determine, by the radiotracer technique, why the crude-oil speed is slower than the speed of water in the HCT pipeline, and (2) to develop anumerical-simulation-based method, using CFD approach, to predict separation length in a turbulent multiphase flow system. For radiotracer practitioners, including experts, this issue is very important in order to provide evidence that the performance of radiotracing is in accordance withthe existing guidelines on the practical applications of radiotracer. As mentioned elsewhere [11,15] and in the procedure adopted here, the distance from the injection point to the first radiation detector has to beat least 50 times theinner diameter of the pipe. The argument is that at that point the fluid flow in the pipeline has fully developed which is a prerequisite for the analysis of RTD curves obtained from the injection of the radiotracers. Consequemtly, the second objective of this study is to verify this fulfillment of this requirement by using CFD simulation.

THEORY

Governing equations

The governing equations for describing multiphase flow are the continuity equation and the momentum equation. The momentum equation, also known as the Navier-Stokes equation, was originally derived from the Newton's second law of motion applied to fluid in laminar condition. It is not easy to solve the Navier-Stokes equation because of thenon-linearity properties of convective term and the presence of the pressure gradient term as a consequence of motion of fluid. Unlike the continuity equation, the pressure gradient term is not stand-alone. A constitutive equation derived from thermodynamics relation is needed to solve pressure gradient equation. In turbulent condition, the flow situations are getting more complex due to the instability of the flow which results in random motions. The Navier-Stokes equation therefore needs to be modified to accommodate turbulent effects. In the turbulent regime, fluctuations around the mean velocity and other variables may occur. These effects need to be incorporated into a CFD model in order for the model to be able to provide meaningful results. This is done through the use of turbulent models, most of which involve a process of time-averaging of conservation equations. Velocity in turbulent flow, for example, is assumed to be the sum of a mean value and a fluctuating component.

In this paper, multiphase flow in HCT pipeline is modeled using the Eulerian-Eulerianthree-fluid model (water, crude oil and gas). The governing equations are adopted from commercial Fluent 6.2 manual for mixture model [16,17]. In Eulerian mixture model gas and liquid phases are treated mathematically as interpenetrating continuum media. The derivation of conservation equations for mass and momentum are performed by ensemble averaging of the local instantaneous flow quantity of each of the phases. In the mixture model water, crude oil and gas are treated as incompressible and are allowed to move with different velocities leading to the use of the term “slip velocity”.

The mass conservation equation for each phase is

$$\frac{\partial}{\partial t}(\rho_i\alpha_i) + \nabla.(\alpha_i\rho_i\vec{u}_i) = 0 \qquad (1)$$

where ρ_i, α_i, and $\vec{u}_i$ represent the density, volume fraction and mean velocity, respectively, of phase i (W = water, CO = crude oil or G = gas). Water, crude oil and gas are assumed to share space in proportion to their volume such that their volume fractions sum is equal to unity in the cells domain

$$\alpha_W + \alpha_{CO} + \alpha_G = 1 \qquad (2)$$

The momentum conservation equation for the phase i after averaging is

$$\frac{\partial}{\partial t}(\rho_i\alpha_i\vec{u}_i) + \nabla.(\rho_i\alpha_i\vec{u}_i\vec{u}_i) = -\alpha_i\nabla p + \nabla.\bar{\bar{\tau}}_{eff} + \rho_i\alpha_i\vec{g} + \sum_{p=1}^{n}\vec{F}_{ij}\left(\vec{U}_{ij} - \vec{U}_{ji}\right) \qquad (3)$$

p is the pressure shared by the three phases and $\vec{F}_{ij}$ represents the inter-phase momentum exchange terms. The Reynolds stress tensor $\bar{\bar{\tau}}_{eff}$ is related to the mean velocity gradients using a Boussinesq hypothesis as expressed the following equation

$$\bar{\bar{\tau}}_{eff} = \alpha_i\left(\mu_{lam,i} + \mu_{t,i}\right)\left(\nabla\vec{u}_i + \nabla\vec{u}_i^T\right) - \frac{2}{3}\alpha_i\left[\rho_i k_i + \left(\mu_{lam,i} + \mu_{t,i}\right)\nabla.\vec{u}_i\right]\bar{\bar{I}} \qquad (4)$$

Turbulence modeling

The turbulence level of the multiphase flow in an HCT pipeline is characterized by intensity of interactions among the fluids flowing inside. The simple way to qualify the turbulence is based on Reynolds number calculation for each phasic flow. As mentioned previously, regardless of the turbulence level, the Navier-Stokes equation needs to be modified to accommodate turbulence effects. The equation known as the Reynolds-Averaged Navier-Stokes (RANS) equation is commonly used to describe fluid flow in turbulent condition. Unfortunately, Reynolds stresses introduce new unknowns to the RANS equation, leading to the number of equations being fewer than the number of unknowns. This is known as the closure problem. To close the problem, additional equations based on turbulent model are needed such that the number of equations and the number of unknown are the same. The turbulent $k-\varepsilon$ model is one of the available turbulent two equation models that can be used to fulfill this requirement. This model is computationally robust and stable even for complex systems.The $k-\varepsilon$ turbulent model is semi-empirical and is based on vast observations of mostly high Reynolds number flow, therefore it is preferable to be usedin engineering calculations. Regarding its name,the turbulent $k-\varepsilon$ model uses two transport equation, namely the kinetic energy turbulence, k, and the rate turbulent dissipation, ε, to compute the Reynolds stresses. The kinetic energy turbulence and the rate dissipation of turbulence are respectively formulated as

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial}{\partial x_i}(\rho U_i k) = \frac{\partial}{\partial x_i}\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i} + G_k - \rho\varepsilon \qquad (5)$$

$$\frac{\partial(\rho\varepsilon)}{\partial t} + \frac{\partial}{\partial x_i}(\rho U_i\varepsilon) = \frac{\partial}{\partial x_i}\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial\varepsilon}{\partial x_i} + C_1\frac{\varepsilon}{k}G_k - C_2\rho\frac{\varepsilon^2}{k} \qquad (6)$$

where

$$G_k = -\rho\overline{U_i'U_j'}\frac{\partial U_j}{\partial x_i} \qquad (7)$$

EXPERIMENTAL METHODS

The experiment was carried out by injecting iodine-based radiotracer into hydrocarbon transport (HCT) pipeline containing multiphase fluids flow. The inner diameter of pipe is 24 in (0.6096 m). The system, consisting of water 95%; crudeoil 3%; and gas 2%, was operated attemperature of 70^{o}C to ensure that fluids were able to flow in the pipeline. The radiotracer iodine-131 was produced in a nuclear reactor and its properties are summarized in Table 1.

Table1. Properties of ^{131}I radionuclide.

Radionuclide	Half life	Gamma energy	Chemical compound	
			Organic	Aqueous
^{131}I	8,04 day	0,36(80%), 0,64(9%)	$^{131}IC_6H_5$	^{131}INa

The selection of radiotracer was based on the compatibility of the radiotracer with the bulk fluids investigated and on the half life and gamma energy ofthe radiotracer. ^{131}I in form of iodobenzene solution ($^{131}IC_6H_5$) was prepared for flow velocity measurement of crude oil because this solution dissolvesin the organic liquid, whereas ^{131}I radiotracer in form of sodium-iodide solution (^{131}INa) was prepared for flow velocity measurement of water because this solution dilutes in aqueous substance. The area of experiment has been chosen at the pipe segment far from branching pointwith assumption that the fluids flow in that areahas stratified and fully developed. Two collimated scintillation detectors denoted by D_1 and D_2 (Ludlum Measurement, USA) had been placed at the distance of 80 and 100 m from injection point, respectively. The experimental set-up is shown in Fig.1.

Fig. 1. Radiotracer experiment on multiphase flow in the HCT pipeline.

Prior to injection, the radiotracer of amount approximately 1 cm^3 was collected into the container of an injector. The radiotracer was then injected successively using the injector which was equipped with an expandable adjustable flexible tubing. High pressure nitrogen was required to push the radiotracers into the pipeline. With this technique, the injection was performed as safely and quickly as possible to producesuitable data of RTD measurement. The iodobenzene solution was injected to measure crude oil flowvelocity, whereas the sodium iodide solution was used to measure water flow velocity. Gas flow was not measured because of unavailability gas tracer.

RESULTS AND DISCUSSION

The residence time distribution (RTD) curves obtained from injection into a multiphase fluids flow are presented in Figs. 2 and 3 for water flow and crude oil flow respectively. The RTD curves generated by scintillation detector represent concentration of the radiotracer in the system with respect to time. A radiotracer solution injected into a fully developed pipe flow disperses because of (i) axial and radial molecular diffusion, (ii) convection, which transports the solute downstream and spreads it as a result of the non-uniform velocity distribution and (iii) gravitational effects, which are the result of density differences between the fluid and the injected solute [18].

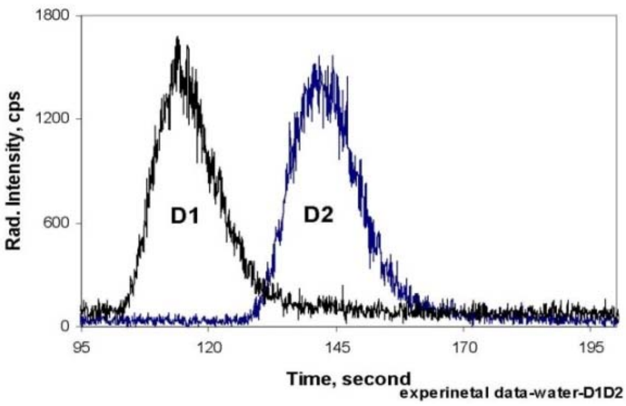

Fig. 2. RTD data obtained from injection ^{131}INa radiotracer solution for measurement of water flow velocity.

Flow velocity calculation

The flow velocity calculation for each phase in a multiphase system depends on the shape of the RTD curves obtained from experiment. Theoretically, two methods can be adopted, namely peak-to-peak and mean residence time (MRT) modes. The peak-to-peak mode is a straightforward method to obtain the flow velocity by measuring the peak positions of the curve. This mode is applicable for situations where the shape of RTD curve is slim, symmetric, and containing one peak only. In real situations, however, the obtained RTD curves often do not show the ideal shape; therefore, the calculation of flow velocity is best performed using the MRT method as done in this paper.

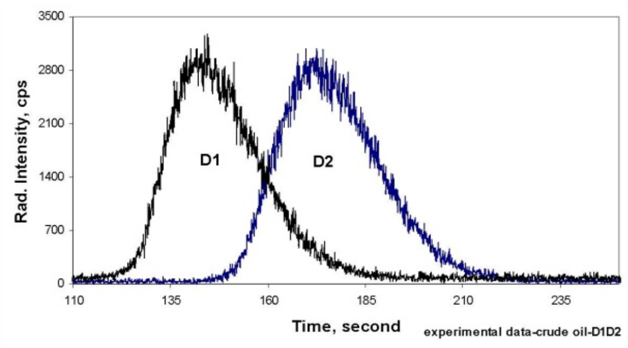

Fig. 3. RTD data obtained from injection $^{131}IC_6H_5$ radiotracer solution for measurement of crude oil flow velocity.

Flow velocity calculations for both water and crude oil are done using the MRT or first moment formula as follows [5,10,19]

$$\bar{t}_i = \frac{\int_0^t t_i C_i(t)dt}{\int_0^t C_i(t)dt} \quad (8)$$

where i= 1 for experimental RTD curve of D_1 and i = 2 for experimental RTD curve of D_2. $C_i(t)$ is tracer concentration at the time t. The denominator of Eq. (8) represents the area under an RTD curve which is proportional to the tracer concentration. The difference in the first moment of the two curves gives the transit time of fluid in the pipeline. Thus:

$$\bar{t} = \bar{t}_2 - \bar{t}_1 \quad (9)$$

As the system was operated normally during the course of experiment time, it was assumed that the system was time-invariant which means that the quantities of flow parameters including the volumetric flow were constant. As the distance of the two detectors is definitely known and the inner diameter of the pipe was fixed, the flow velocities of water and crude oil can be calculated precisely. The results are summarized in Table 2.

Table 2. Transit time calculation for flow velocities of water and crude oil respectively.*) The detector position is measured from injection point.

Liquids	Mean Residence Time (s)		Detector position*		Transit time (s)	Flowvelocity (m/s)
	D1	D2	D1	D2		
Water	119.195	146.187	80	100	26.992	0.74096
Crude Oil	148.913	178,053	80	100	29.140	0.68651

The results of calculations show that the water flow velocity is higher than crude oil flow velocity, even though the density of the water is higher than the density of crude oil. This important result is probably best explained by the fact that the system under investigation is water-dominated. The water content in the pipeline, which is 95%, is much bigger than crude oil and gas contents which are 3% and 2% respecively. In a water dominated system, the water can serve as carrier due to its higher momentum generated from pumping force. Additionally, in this closed system, the crude oil movement is slowed by frictions both with gas at top layer and with water at the water-crude oil interface.

The turbulence level of the liquid is predicted by using the Reynolds Number formula, Re, through the following relation [20]

$$Re = \frac{ud}{\nu} \quad (10)$$

whereRe is the Reynolds Number, which is dimensionless.As previously mentioned, the system under investigation is water-dominated; therefore, to apply Eq. (10), it is assumed the pipe isfilled with water. The physical properties of waterat the temperature of 70°C are obtained from the literature [20], as follows: density ρ = 978 kg/m^3, kinematic viscosity ν = 4.11×10^{-7}m^2/s, and average velocity of flow u (calculated from MRT) = 0.74096m/s and inner diameter of the pipe $d = 0.6096\,m$. The rough estimation of Reynolds number calculatedby Eq. (10) resulting Re = 987406 which indicates that the water flow in pipeline is turbulent.

Prediction of separation length

The separation length was predicted using Fluent mixture procedure of CFD simulation. Here the term “separation length” refers to the length from which the mixed fluids at injection point is decomposed into its stratified phasic flows.Here we assumed that the stratifiedflows were due to following affecting factors: (1) gravitational force acting on whole fluids, (2) slip velocity due to differing velocities of phasic flow, (3) differences inviscosities and (4) differences in volume fraction of each fluid.

Method of solution

The CFD simulation is used to obtain the numerical values of the flow variable, ϕ, as a function of space and time at the grid points of the computational domain of the investigated system. The variable ϕ can stand for pressure p, velocity components u_i, or temperature T. Its value is found by iterative solution of the set of algebraic equations of the form

$$a_P \phi_P = \sum_{nb} a_{nb}\phi_{nb} + C_{sb} \tag{11}$$

where a_P is the center coefficient, $a_{nb}$is the influence coefficient for the neighbor, and C_{sb} is the contribution of the constant part or the source term S_C in $S = S_C + C_{ps}$ and the boundary conditions. Eq. (11) is derived from the discretization procedure to governing equations using finite volume method (FVM) proposed by Patankar [21].

The first step of the procedure for solving the governing equation is to convert the physical domain into the computational domain. In this regard, the physical domain is the segment of pipeline of a length of 80 m from injection point to the point on which the first radiation detector is placed. Uniform grid of meshing on computational domain was performed by implementing triangular control volumes using the Gambit 2.2 mesh generator [16]. Due to the symmetry of the pipelines viewed from $x - y$ coordinate system and computational speed reasons, the meshing map was defined in 2D. A sample of the triangular meshing is presented in Fig. 4.

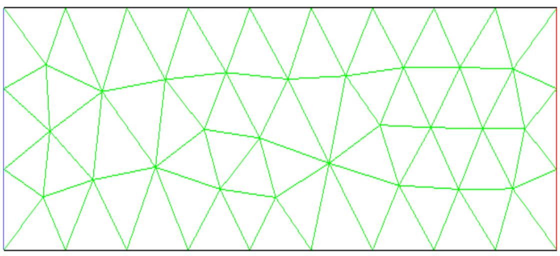

Fig. 4. A sample of the meshing map on computational domain using triangular control volume

The robustness of an iterative process depends on the effectiveness and stability of the grid-generation scheme employed for investigation. In the present work all prerequisite for desired solution has been done in Fluent solver. A first-order upwind scheme was used to discretize the momentum equations, volume fraction, turbulent kinetic energy, and turbulent dissipation energy. This scheme was employed to ensure satisfactory accuracy, stability and convergence. The boundary conditions were specified based on Fluent documentation [16]. The inlet turbulent kinetic energy, k, was estimated from turbulence intensity as expressed

$$k = \frac{3}{2}(uI)^2 \tag{12}$$

where I is the turbulence intensity being given by

$$I = 0.16\left(Re_{d_H}\right)^{-1/8} \tag{13}$$

The inlet turbulent dissipation rate, ε, was estimated from the turbulent viscosity ratio as expressed by Eq. (14)

$$\varepsilon = \rho C_\mu \frac{k^2}{\mu}\left(\frac{\mu_t}{\mu}\right)^{-1} \tag{14}$$

where C_μ is an empirical constant specified in the turbulence model (0.09). During the simulations of turbulent multiphase flow, it was employed standard wall functions available in the commercial CFD solver. In the CFD multiphase simulations, the SIMPLE (Semi Implicit Method for Pressure Linked Equation) algorithm was used for the pressure-velocity coupling [21]. In this algorithm, the new velocities are computed based on guessed pressure field in momentum equations in a segregated fashion, but these will not, in general, satisfy the continuity equation, so corrections to the velocities are determined. Based on the velocity corrections, a pressure correction is computed which, when added to the original guessed pressure, result in an updated pressure. The process is repeated until corrected pressures and corrected velocities satisfy the continuity equation. All predetermined numerical solution parameters, including initial and boundary conditions, are summarized in Table 3.

Table 3. Predetermined numerical solution parameters for CFD simulation

Phases	Water, ρ= 988.2 kg/m^3, vol. frac = 0.95 Crude oil, ρ= 780 kg/m^3, vol.frac = 0.03 Gas, ρ= -, volume fraction = 0.02
Meshing:	uniform grid, control volume: triangle number of node : 11799
Boundary conditions:	velocity magnitude (water): 0.74 m/s velocity magnitude (crude oil): 0.69 m/s velocity magnitude (gas): 0 m/s Turbulent intensity: 10% (%) Turbulent length scale: 0.025
Operating condition:	gravity: -9.81 (m/s^2)
Turbulent model:	k-epsilon (standard) standard wall function
Control solution:	flow volume fraction slip velocity SIMPLE Under relaxation
Residual:	10^{-5}-10^{-3}

The criterion of convergence is based on the predetermined residual value of the calculated variables, ϕ, in equations of continuity, velocity components, turbulent kinetic energy, turbulent dissipation energy and volume fractions. Solution of these equations is considered to have converged when the following condition is satisfied [1]

$$\frac{R_\phi^n}{R_\phi^m} \leq 10^{-5} - 10^{-3} \qquad (15)$$

where R_ϕ^n and R_ϕ^m denotes residual value of the variable ϕ after n and m iterations respectively.

The convergence condition for current work was achieved after 4500 iterations as presented in Fig. 5. All solved equations in the simulation are also shown.

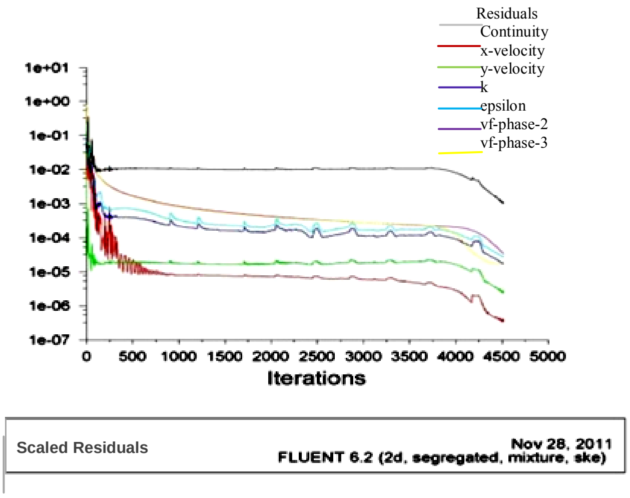

Fig. 5. Iterative process for convergence test in multiphase flow simulation.

The predicted separation length in the multiphase flow is shown in Fig. 6 for 2D presentation. The presentation of the figure is inthe format of volume fraction, because this format is the most suitable in representation of separation length. As previously mentioned, the separation process inthis simulation was affected by following factors: gravity, differences in density, viscosity, and velocity of each phase respectively, as summarized in Table 3. As can be seen from Fig. 6, full separation process, which leads to stratified flows, approximately occurred at the distance of 20 m from injection point. Fortunately, the stratified flow can be presented clearly because water, crude oil and gas are immiscible with each other.

From a practical point of view, theresults obtained from the current work are useful because they can serve as a starting point to attain a new insight in predicting flow process in general and separation length of stratified flow in a multiphase fluidic system in particular. An extension of this result can be produced by conducting similar works of multiphase flows for various pipe diameters. As mentioned previously, the placement of the first radiation detector from injection point in radiotracer work is at the distance of at least 50 times pipe diameter. In the current work the distance to the first detector from injection point is 80m, which exceeds the minimum distance prerequisite (50 × 24 in = 30,48 m)and separation length (20m) as well. In other words,the placement of the detector in this workwas correct.

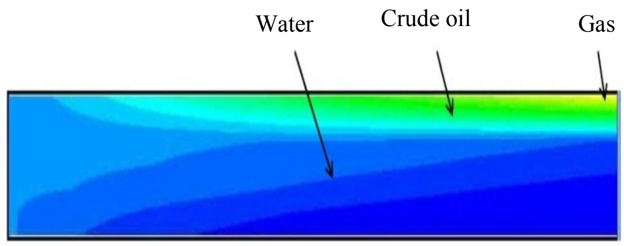

Fig.6. The prediction of separation length in multiphase flow simulated by mixture model of CFD simulation. Stratified flow was predicted occurred at 20 m from injection point (note: the scale of the figure is not proportional)

CONCLUSION

An experiment with radiotracer and CFD simulations using mixture model have been successfully conducted to study prediction of separation length of multiphase fluid flows in HCT pipeline with a diameter of 24 in. The conclusions drawn from this study are as follows. First, in a water dominated system, the flow of water is faster than the flow ofcrude oil despite the higher density of water than the density of crude oil. Second, the predicted separation length of immiscible fluids (water, crude oiland gas) in the multiphase flows is approximately 20 m, measured from injection point. And third, the placement of the first detector at the distance 80 m from injection point was correct as required for practical radiotracer work.

ACKNOWLEDGMENT

The authors acknowledge the support of the Polish Ministry of Science and Higher Education. This work was financially supported by Indonesian Ministry of Research and Technology under Ph.D Scholarship Program and IMHERE B.2C Bandung Institute of Technology Sandwich Doctoral Program 2011.

REFERENCES

1. S. Ghorai and K.D.P. Nigam,Chem. Eng. Process.**45** (2006) 55.
2. C.H. Newton and M. Behnia, Int. J. Multiphase Flow **26** (2000) 327.
3. P.A. Berthelsen and T. Ytrehus, Int. J. Multiphase Flow **31**(2005) 571.
4. Mr. Piput, Personal Communication, PT. BumiSiakPusako, Jakarta (2007).
5. O. Levenspiel, Chemical Reaction Engineering, 2nd ed., John Wiley & Sons., New York (1972) 253.
6. L. Furman, J.P. Leclerc and Z. Stegowski, Chem. Eng. Sci.**60** (2005) 3043.
7. P.V. Danckwerts, Chem. Eng. Sci. **2** (1953) 1.
8. J. Thŷn, R. Zitny, J. Kluson *et al*,Analysis and Diagnostics of Industrial Processes by Radiotracers and Radioisotope Sealed Sources, ČVUT., Praha (2000).
9. J.P. Leclerc, B. Antoine, S. Claudel *et al.*,Oil and Gas Science and Technology Rev. IFP 55 (2000) 159.
10. Anonymous, Radiotracer Residence Time Distribution Method for Industrial and Environmental Applications, Training Course Series 31, IAEA, Vienna (2008).
11. P.A.A. Klusener, G. Jonkers, F. During*et al.*, Chem. Eng. Sci. **62**(2007) 5495.
12. Le Moullec, O. Potier, C. Gentric*et al.*, Chem. Eng. Sci. **63** (2008) 2436.
13. L. Furman and Z. Stegowski, Chem. Eng. Process.**50** (2011) 300.
14. S. Sugiharto, Z. Su'ud, R. Kurniadi, W. Wibisono and Z. Abidin, J. Appl. Rad. Isot. **67** (2009) 1445.
15. A.E. Hills, Practical Guidebook for Radioisotope-Based Technology in Industry, IAEA/RCA/RAS/8/078, Vienna (1999) 55.
16. Anonymous, Fluent Users Guide, Version 6.2, Fluent Inc., Beirut (2006) 89.
17. E.M. Marshal and A. Bakker, Computational Fluid Mixing, Fluent. Inc., Beirut (2002) 17.
18. E. Ekambara and J.B. Joshi, Chem. Eng. Sci. **58** (2003) 2715.
19. J.A. Niemi, Int. J. Appl. Rad. Isot.**28** (1977) 855.
20. R.L. Mott, Applied Fluid Mechanics, 5th ed., Prentice Hall. Inc. Pearson Education, New Jersey (2000).
21. S.V. Patankar, Numerical Heat Transfer and Fluid Flow, Hemisphere Publishing Coorporation, Taylor and Francis Group, New York(1980).