

CFD Prediction of Oil-Water Phase Separation in 180° Bend

Mohammed A. Al-Yaari* and Basel F. Abu-Sharkh

Department of Chemical Engineering, King Fahd University of Petroleum & Minerals, Saudi Arabia

* Corresponding author; E-mail address: malyari@kfupm.edu.sa

Abstract—Oil-water two-phase flow in 180° bend is numerically simulated using FLUENT 6.2. A flow in 150, 300, 600 mm radius bends with a circular cross-sectional area (100 mm ID) are simulated using Eulerian–Eulerian approach. Standard k-ε turbulence model is adopted. Three different inlet velocities (0.3, 1 and 3 m/s) are studied for all the three bends. The phase separation and turbulent flow structure are investigated for the nine test matrix. The results show negative effect, on oil-water phase separation and turbulent mixing inside the studied 180° bends, as inlet velocity and/or bend to pipe radius (R/r) ratio increases.

Index Terms— 180° bend flow, Bend radius effect, CFD simulation, Inlet velocity effect, Oil–water flow, Turbulent flow

I. INTRODUCTION

THE existence of bends is absolutely necessary in piping Systems. Bends play a role in developing flexibility in a system that transports a fluid or solid material from one place to another. Besides flexibility, a pipe bend with a specific shape, i.e. a U-bend, provides compactness and effectiveness for the purpose of transferring heat. The compactness and effectiveness of a U-bend makes it appropriate in heat exchangers, cooling ducts and pneumatic conveying dryer applications.

Return bends are curved pipe fittings which connect parallel straight tubes in finned-tube heat exchangers, such as evaporators and condensers used in air conditioning and refrigeration systems.

Manuscript received October 6, 2011. This work was supported by the National Science, Technology and Innovation Plan (NSTIP) under project No. 09-OIL 788-04.

Mohammed A. Al-Yaari is with Chemical Engineering Department, King Fahd University of Petroleum & Minerals, Dhahran, Saudi Arabia (corresponding author to provide phone: +966-3-860-7279; fax: +966-3-860-4234; e-mail: malyari@kfupm.edu.sa).

Basel F. Abu-Sharkh is with Chemical Engineering Department, King Fahd University of Petroleum & Minerals, Dhahran, Saudi Arabia (e-mail: Sharkh@kfupm.edu.sa).

A strong demand for the analysis of multiphase flow problems is felt in the field of process engineering. Specific stimulus comes from the oil and gas production industry.

Specially, bends are a common element in any piping system of oil-water flow applications such as petroleum production, transportation and some petrochemical industries. The oil-water flow patterns in bends are affected by complex parameters, such as centrifugal forces, and secondary flows. A basic understanding of liquid-liquid flow in a U-bend is frequently required to obtain better U-bend design for such fluids.

Benedetto Bozzini et al. [2] used CFD to simulate a typical operating condition for off-shore oil extraction industry implying pipe flow of two immiscible liquids (oil and sea water), one gas (hydrocarbon mixture) and one dispersed solid (sand) in 90° elbow. They tried to evaluate the erosion-corrosion issues for such system.

Pitor A. Domanski and Christian J. L. Hermes [6] proposed a correlation for two-phase flow pressure drop in 180° return bends based on a total of 241 experimental data points from two independent studies. They used smooth tubes with inner diameters (ID) from 3.3 mm to 11.6 mm, bend radiuses (R) from 6.4 mm to 37.3 mm, and curvature ratios (2R/D) from 2.3 to 8.2. The correlation consists of a two-phase pressure drop for straight-tubes and a multiplier that accounts for the bend curvature.

K. Ekambara et al. [3] used the volume averaged multiphase flow equations to model the internal phase distribution of co-current, air-water bubbly flow in a 50.3mm ID horizontal pipeline. They studied the effect of liquid and gas volumetric superficial velocities and average gas volume fraction. They argued that k-ε model with constant bubble size and k-ε with population balance model showed better agreement with the experimental data with population balance than the constant bubble size predictions. In addition, they reported that the volume fraction has a maximum near the upper pipe wall, and the profiles tend to flatten with increasing liquid flow rate. Furthermore, they reported that axial liquid mean velocity showed a relatively uniform distribution except near the upper pipe wall and the liquid velocity distribution tended to form a

fully developed turbulent pipe-flow profile at the lower part of the pipe irrespective of the liquid and gas superficial velocities.

Arlindo de Matos and Fernando A. Franca [1] got experimental and numerical data on the phase distribution of gas-liquid bubbly flows taking place inside a pipe of square cross-section. They focused on the phase segregation that happens when the mixture goes through a U-bend curve. Their set-up was connected to run these air-water flows at nearly atmospheric pressure. They reported that along the U-bend curve, the gas bubbles were displaced to the inner curve section caused by the action of centrifugal fields. Also, they reported that the action of the centrifugal field, setting the phase distribution along the curve, could be measured and modeled by using the time-averaged Two-Fluid Model.

Samy M. et al [7] simulated gas-solid two-phase flow in 180° curved duct is Eulerian-Lagrangian approach. They adopted four turbulence models namely; standard $k-\epsilon$ model, RNG (Renormalization Group) based $k-\epsilon$ model, Low-Re $k-\epsilon$ model and an extended version of the standard $k-\epsilon$ model. They reported that the RNG based $k-\epsilon$ model predicts the flow behavior better than other models. In addition, they focused on the effects of inlet gas velocity, bend geometry, loading ratio and particle size on the flow behavior and bend pressure drop. They got some results showing that the flow behavior is greatly affected by these parameters.

The studies of oil-water flow in pipes have attracted more attention than in bends, especially in 180° bend. Therefore, the aim of the present study is to investigate numerically the flow of oil-water in 180° curved pipe in order to understand the physical phenomena of such flow in such geometries. Furthermore, phase separation, inlet velocity and bend to pipe radius (R/r) ratio effects are studied numerically.

II. MATHEMATICAL MODEL

The commercial FLUENT software package, FLUENT 6.2, was used for solving the set of governing equations. The numerical method employed is based on the finite volume approach (Fluent, 2001). Fluent provides flexibility in choosing discretization schemes for each governing equation. The discretized equations, along with the initial condition and boundary conditions, were solved using the segregated solution method to obtain a numerical solution.

The Eulerian model is employed to predict the oil-water flow behaviour at a U-bend. In the Eulerian method, two phases are modelled as two inter-penetrating continuous media. The first phase is called the continuous phase and the second one is normally called the dispersed phase. The quantities of continuous and dispersed phase in the system are represented by their volume fractions. Both phases are linked in the momentum equation. The standard $k-\epsilon$ model, derived by assuming fully turbulent flow and negligible effects of molecular viscosity, was used to model turbulence phenomena in both phases.

In the region near the wall, the gradient of quantities is considerably high and requires fine grids close to the wall to capture the change of quantities. This causes the calculation to become more expensive meaning time-consuming, requiring greater memory and faster processing on the computer, as well as expensive in terms of complexity of equations. A wall function, which is a collection of semi-empirical formulas and functions, provides a cheaper calculation by substituting the fine grids with a set of equations linking the solutions' variables at the near-wall cells and the corresponding quantities on the wall.

III. NUMERICAL SOLUTION

A. Geometry

Figure 1 shows a 2D sketch of a 180° return bend connecting two parallel straight-tubes. The flow patterns of two-phase flow in a straight-tube, whose characteristics depend on different parameters such as diameter, pipe roughness, pipe material, velocity, physical properties, ...etc (M. Al-Yaari, et al [5]).

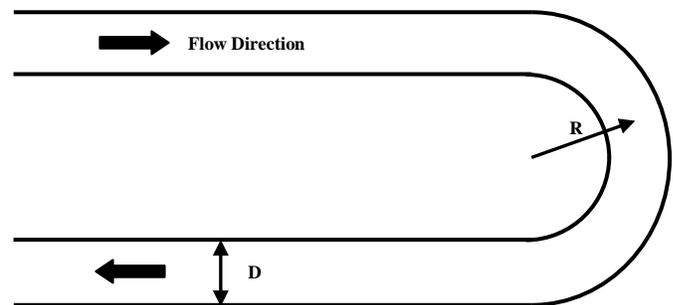


Figure 1 2D Schematic view of 180° curved pipe.

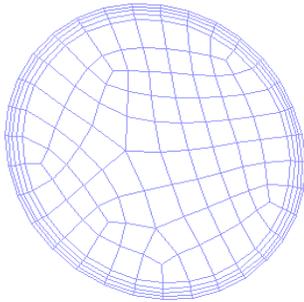
A sketch of the geometry of the calculation domain is shown in Figure 2. The geometry consists of three parts, i.e. the upstream pipe, the U-bend and the downstream pipe. The length of the upstream and downstream pipes is 10 times the diameter of the pipe. The former is used to reach both a steady, fully developed flow. The latter is used to avoid possible recirculation flow paths at the outlet surface of the domain, thus leading to numerical convergence errors or unphysical results. The diameter of the pipe for the present work is 0.1 m while the bend to pipe radius ratios are 3, 6 and 12.

The computational grid of 53599, 63581 and 83762 cells were generated and used. The grid was generated using Gambit 2.2, which is compatible with Fluent 6.2.

A boundary layer, which contains four cells with a distance of the cell adjacent to the wall at 1 mm, and the growth factor of 1.2, is employed on the wall to improve the performance of the wall function and to fulfill the requirement of y^+ , the dimensionless wall distance, for the cell adjacent to the wall which is in the range 50–500 (Fluent, 2001).. The dimensionless distance y^+ is defined by:

$$y^+ = \frac{\rho u_\tau y_p}{\mu}$$

To obtain better convergence and accuracy for a long pipe, the hexagonal shape and Cooper-type elements have been employed. The Cooper-type element is a volume meshing type in Gambit, which uses an algorithm to sweep the mesh node patterns of specified 'source' faces through the volume.



(a)



(b)

Figure 2 (a) Mesh system for the cross-sectional area of circular pipe.
(b) Mesh system for the longitudinal plane of the 180° bend

The space domain for the CFD analysis refers to a 180° bend, 100 mm ID pipe. A three-dimensional mesh has been set up, by adding further volumes both at the inlet and the outlet of the bend.

The adoption of a three-dimensional geometry is a mandatory choice, in order to take into account possible effects of secondary flow paths. These secondary flow paths develop in transverse planes with respect to the main flow direction and are well known fluid dynamic characteristic of flow in bends, arising from centrifugal effects on the fluid due to the curvature of the domain. Moreover, gravitational effects could be difficult to account for in two-dimensional geometry.

B. Boundary Conditions

There are three faces bounding the calculation domain: the inlet boundary, the wall boundary and the outlet boundary. Flat velocity profile and oil and water volume fractions of 0.7 and 0.3 were introduced at the inlet of this section. The outlet boundary condition of the latter was set up as a pressure outlet boundary. No slip was used to model liquid velocity at the wall. The main physical properties for the fluid phases are reported in Table 1.

Table 1 Main physical properties for the fluid phases

Property	Water Phase	Oil Phase
Density (ρ), kg/m ³	998.2	780
Dynamic Viscosity (μ), Pa.s	0.001003	0.00157

Two fluid dynamic characteristic parameters have been selected as key points for the case matrix definition, namely: (i) fluid flow inlet velocity, (ii) bend pipe radius ratio.

Three values for each parameter have been selected to compose the nine cases set, representing typical high, medium and low levels of the relevant quantities, in a design of experiments frame of mind. The resulting case matrix is reported in Table 2.

Table 2 Case matrix for the CFD analysis

Parameter	Low	Medium	High
Inlet Velocity, m/s	0.3	1	3
Bend Pipe Radius Ratio (R/r)	3	6	12

C. Solution Strategy & Convergence

A first-order upwind discretization scheme was used for the momentum equation volume fraction, turbulent, kinetic and turbulent dissipation energy. This scheme ensured, in general, satisfactory accuracy, stability and convergence. In addition, the steady-state solution strategy was employed.

The convergence criterion is based on the residual value of the calculated variables, i.e. mass, velocity components, turbulent kinetic energies, turbulent dissipation energies and volume fraction. In the present calculations, the threshold values were set to a ten thousandth for continuity and a thousandth for the remaining equations. These values are considered small enough to produce accurate results.

Other solution strategies are: the reduction of under-relaxation factors of momentum, volume fraction, turbulence kinetic energy and turbulence dissipation energy to bring the non-linear equation close to the linear equation, subsequently using a better initial guess based on a simpler problem.

IV. RESULTS AND DISCUSSIONS

The oil-water flow in 180° bends has been studied numerically. To investigate the effect of inlet velocity on the oil-water phase separation, three different inlet velocities (0.3, 1 and 3 m/s) have been tested. In addition, the effect of bend to pipe radius ratio on oil-water phase separation in a U bend, has been achieved by using three different R/r ratios (3, 6 and 12).

A. Oil-Water Phase Separation

1. Effect of inlet velocity

The resultant data of inlet velocity effect on the oil-

water phase separation are presented in Table 3 below. As shown in that table, for all the studied R/r ratios, as the inlet velocity increases, the oil-water phase separation inside U bend components decreases. For the case where R/r = 3, at very small inlet velocity, there is enough time for the flow to show phase separation. However, this could be because the oil-water system with such physical properties may not exist in the dispersed phase at such small inlet velocity and this is in a good agreement with M. Al-Yaari et al. (2009) results. For the case of higher values with same R/r value, lower separation has been detected and this could be attributed to small length of the downstream pipe, which results in circulation flow (see Figure 4) and the probability of such system to exist in the dispersed flow pattern in such conditions. The oil volume fraction contours for all the three cases (with inlet velocity of 0.3, 1 and 3 m/s) when R/r = 3 are shown in Figure 3. Further research should be achieved using other multiphase models such as volume of fluid to model such system at very small inlet velocity.

2. Effect of R/r ratio

As shown in Table 3 above, as the bend to pipe radius ratio increases, the tendency of oil-water system to show phase separation decreases. Furthermore, at higher R/r values (6 and 12), almost no phase separation occur. Such results are logic and can be attributed to either the short length of the downstream pipe or the high velocity at the inlet section.

Table 3 Effect of inlet velocity and R/r ratio on oil-water phase separation

R/r ratio	Inlet Velocity, m/s	Oil Volume Fraction Range
3	0.3	0.97 – 0.165
	1	0.896 – 0.447
	3	0.77 – 0.606
6	0.3	0.707 – 0.694
	1	0.703 – 0.699
	3	No Change
12	0.3	No Change
	1	No Change
	3	No Change

B. Flow Structure

Effect of inlet velocity on the U bend flow structure of oil-water system has been studied numerically using three different velocities (0.3, 1 and 3 m/s). Moreover, the effect of R/r ratio has been studied for three different R/r ratios. As shown in Fig. 4 below, the turbulent flow structure of oil phase in oil-water system does not show the secondary flow which create vortex inside the 90° elbow for single phase flow as known. With increasing the turbulent intensity, by increasing

Re resulted from increasing the flow velocity, mixing inside 180° bend decreases (see Figure 5).

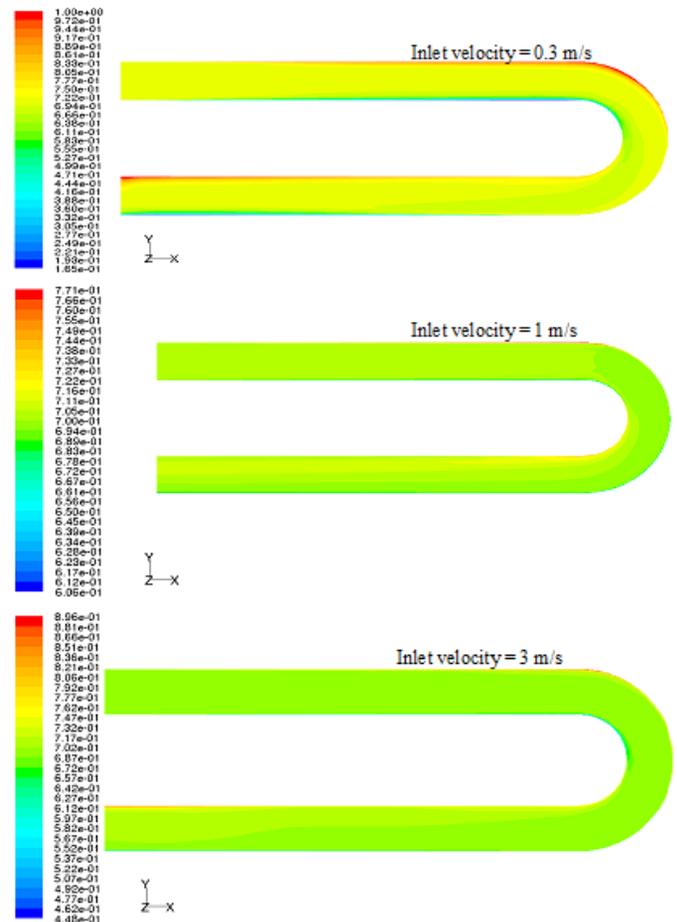


Figure 3 Oil volume fraction contours for R/r = 3

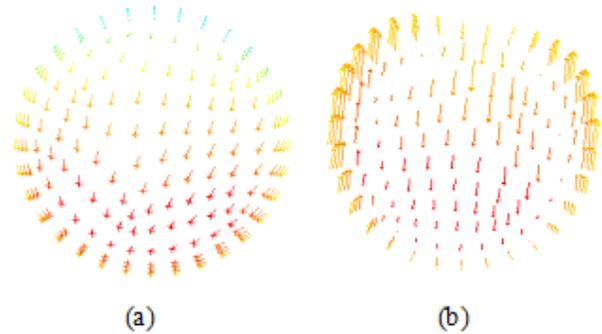


Figure 4 Oil radial velocity vectors (a) at 0.3 m/s when R/r=3, (b) at 3 m/s when R/r=6

However, CFD simulation results indicate that with increasing bend to pipe radius ratio, turbulent mixing inside the 180° bend decreases.

According to Hoang and Davis [4] the two-phase flow in the bend is affected not only by the secondary flow effects observed in single-phase flows, but also by the separation of the phases due to centrifugal forces which concentrates the dense liquid toward the bottom portion, while the lighter one

flows toward the top portion. This increases the relative motion between the phases and pressure drop. At the bend outlet, significant pressure drop is also caused by the remixing process, which extends to about 9 diameters downstream.

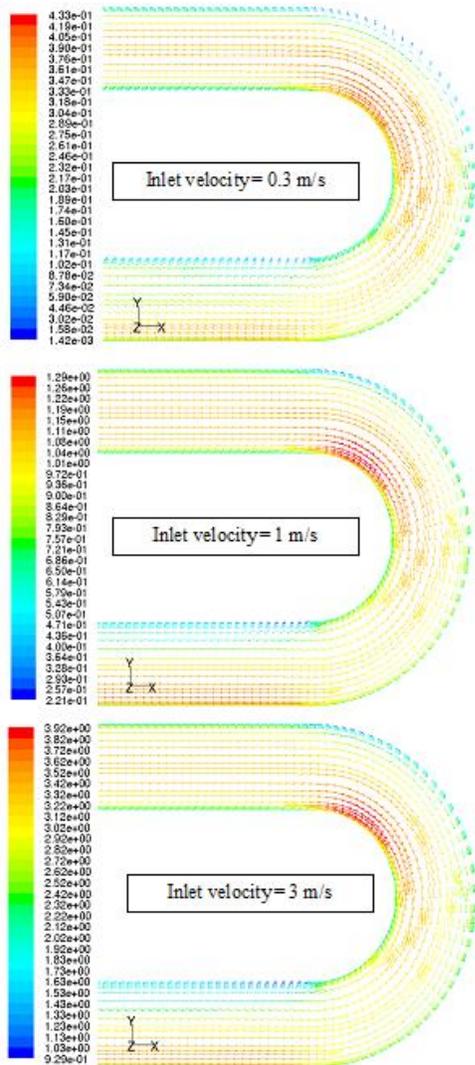


Figure 5 Oil Velocity Vectors for $R/r = 3$

V. CONCLUSIONS

The following conclusive remarks result from our analysis. As far as the fluid dynamic analysis is concerned:

1. CFD calculations using Fluent 6.2 were performed to investigate the influence of a U-bend on the phase separation of oil and water phases and turbulent flow structure of such system using Eulerian-Eulerian multiphase model and standard $k-\epsilon$ two equations turbulent model.
2. U-bend causes an important phenomenon to occur: an accumulation of dense liquid (water in this study) along the outer walls of the U-bend and the beginning of the downstream pipe. This will bring to mind the

corrosion issue caused by salty water in oil production pipelines.

3. Gravitational settling is the main effect at low velocity values, while the centrifugal force is more important at high values.
4. As inlet velocity and/or R/r ratio increases, oil-water phase separation decreases.
5. Oil-water phase separation is no clearer with increasing R/r .
6. The two-phase flow in the bend is affected by the separation of the phases.
7. As inlet velocity and/or R/r ratio increases, more length for the downstream pipe is needed to avoid recirculation.

ACKNOWLEDGMENT

Authors are also grateful for useful discussions with Dr. Rached Ben-Mansour of Mechanical Engineering Department, King Fahd University of Petroleum & Minerals, Dhahran, Saudi Arabia.

REFERENCES

- [1] Arlindo de Matos and Fernando A. Franca, "Bubbly flow segregation inside a U-bend pipe: Experimentation and numerical simulation" *Chemical Engineering Research and Design*, 2009, 87, 655-668.
- [2] Benedetto Bozzini, Marco E. Ricotti, Marco Boniardi and Claudio Mele, "Evaluation of erosion-corrosion in multiphase flow via CFD and experimental analysis", *Wear*, 2003, 255, 237-245.
- [3] K. Ekambara, R. S. Sanders, K. Nandakumar and J. H. Masliyah, "CFD simulation of bubbly two-phase flow in horizontal pipes" *Chemical Engineering Journal*, 2008, 144, 277-288.
- [4] K. Hoang, M.R. Davis, "Flow structure and pressure loss for two phase flow in return bends", *Trans. ASME*, 1984, 106, 30-37.
- [5] M. Al-Yaari, A. Soleimani, B. Abu-Sharkh, U. Al-Mubaiyeh and A. Al-Srakh, "Effect of drag reducing polymers on oil-water flow in a horizontal pipe. International Journal of Multiphase Flows", 2009, 35, 516 - 524.
- [6] Pitor A. Domanski and Christian J. L. Hermes, "An improved correlation for two-phase pressure drop of R-22 and R-410A in 180° return bends", *Applied Thermal Engineering*, 2008, 28, 793-800.
- [7] Samy M. El-Beheery, Mofreh H. Hamed, M. A. El-Kadi and K. A. Ibrahim, "CFD prediction of air-solid flow in 180° curved duct", *Powder Technology*, 2009, 191, 130-142.



Mohammed A. Al-Yaari was born in Sana'a, Yemen, on August 25, 1977. He received the B.Sc. degree in Chemical Engineering from Baghdad University, Baghdad, Iraq, in June 2000 and the M.Sc. degree in Chemical Engineering from King Fahd University of Petroleum & Minerals (KFUPM), Dhahran, Saudi Arabia, in June 2008. He is a PhD Candidate in the Chemical Engineering Department in KFUPM.

His research interest includes drag reduction by polymer additives, CFD simulation and emulsion technology.