A COMPUTATIONAL STUDY OF CURVATURE EFFECT ON PRESSURE DROP OF GAS-LIQUID TWO-PHASE FLOW THROUGH 90 DEGREE ELBOW

by

Hindren Ali SABER[a] and Iyd Eqqab MAREE[b*]

[a] Mechanical and Energy Engineering Techniques Department,
Erbil Technical Engineering College, Erbil Polytechnic University Erbil, Iraqi Kurdistan, Iraq
[b] Aviation Engineering Department, Engineering College,
Salahaddin University Erbil, Iraqi Kurdistan, Iraq

Original scientific paper
https://doi.org/10.2298/TSCI210322002S

The CFD analysis of air-water two-phase flows was performed in a 90º vertical to horizontal elbows made of PVC with four different inside diameters (25 mm, 50 mm, 75 mm, and 100 mm) with six different curvature radius to diameter ratios (R/D) 1, 2, 4, 6, 8, and 10. Pressure drops were investigated at various upstream and downstream locations using computational, experimental, empirical methods. The most effective method in investigating and determining total pressure drop of two-phase flow in pipes and bends is considered to be computational study. The CFD simulations were performed using ANSYSY 19.2 FLUENT and a mixture model. The studies were conducted under the following two-phase conditions: mass quality from 1-50% and mass flux from 350-1000 kg/m²s. The results show that the impact of these significant parameters are important and dramatic specially at high curvature radius, mass flux and mass quality. The results of the CFD study demonstrated a substantial loss in energy and pressure as the fluid exits the elbow section, and also a higher drop in pressure observed at higher air velocity. Also, higher pressure drops were obtained with increasing pipe diameter in one side, and with decreasing R/D on the other side. Finally, the current results were verified with the empirical and experimental studies and a good agreement were obtained.

Key words: *CFD, curvature radius, 90 degree elbow, pressure drop, two-phase flow, vertical riser*

Introduction

Multi-phase flow in fluid mechanics is the continuous flow of substances with two or even more thermodynamic phases. The simple principle of multi-phase flow in pipes and tubes is a two-phase flow that complies with all the basic laws of fluid mechanics. The two-phase flow equations are comparatively complex than the one-phase flow equations. The accurate flow pattern with low energy loss that can be created in gas-liquid framework is one of the major complications. The study and analysis of fluid or particle flow in the elbow is necessary for a broad range of engineering applications, such as chemical processing industry, heat exchanger devices, plumbing and pipe-line systems, aerospace, automotive, air conditioning and refrigeration systems, *etc.* Among the various applications, pipe fittings or elbows are typically used in heating, ventilating and cooling systems for heat transfer processes. Pressure and velocity

* Corresponding author, e-mail: iyd.maree@epu.edu.iq

distributions of the fluid-flow at different positions of the pipe-line and duct systems are critical and limited for efficient heat transfer, mixing and circulation. The pressure drop prediction of the single phase with appropriate accuracy for bend and curved pipe is easy [1]. On the other hand, the prediction of the total two-phase pressure drops and flow type in pipe bends and elbows is complicated, because of the variation in the different parameters namely viscosity, fluid density, phase transition and interactions of the fluids and particles with the inner pipe wall.

The two-phase flow equations compared to the single-phase flow are quite complicated on account of the presence of various flow patterns and its types in multi-phase flow systems [2]. Also, reference [2] stated that the overall pressure drops in the internally waved pipe bends are nearly 2-5 times larger than in the smooth pipe bends and higher friction factor observed due to the intensive circulation in the pipe. Wallis [3] offered a thorough report of the behavior of the two-phase flow phenomenon. The flow patterns noticed in the vertical straight pipe are slug, plug, bubble and wispy annular. While in the horizontal pipe, the observed flow patterns are annular, stratified, bubble and slug. Many endeavors have been made for determining pressure drop in different kinds of flows including the horizontal pipe flow [4], vertical upward flow [5], and pressure loss and friction factor of the two-phase and multi-phase flows [6]. Extensive studies of the pressure drops were largely restricted to the horizontal-to-horizontal same plane. Chenoweth and Martin [7] demonstrated that the higher pressure drop obtained from two-phase flow through the bends were compared to single-phase pressure drop in the same bend, they could be associated with the acceptance of Lockhart and Martinelli [8] theory originally developed for vertical and horizontal straight pipe and to predict pressure drop per unit pipe length in the bend for single-phase gas only. The correlation that they reported can be used to calculate energy losses in the fittings, pipes, elbows and bends. Chisholm [9] developed a rudimentary model for calculating two-phase flow in pipe fittings, based on a factor that called liquid two-phase multiplier, for various geometries and operation conditions such as pipe diameters, curvature radius, and flow rates. Arun *et al*. [10] provided accurate two-phase pressure drop model for pipes and developed an approach for flows of two immiscible fluids are existed in pipes and tubes. Sekoda *et al*. [11] developed a correlation of two-phase multiplier, they reported that the model created in the bend assumed to be a liquid only (single-phase pressure drop). The two-phase pressure drop of the pipe fittings elbows or bends was considered to depend on the curvature radius and to be independent of inner pipe diameters and lengths.

A number of studies on the orientation of the bend plane and surface have always yielded opposite results. So far, the issue of the real influence of pipe bends orientation on two-phase flow energy loss and pressure drop has not been decided. Some authors [12, 13] stated that, there is no real effect of pipe bending orientations on two-phase flow energy and pressure losses. While there are those who acknowledge the existence of this effect [14, 15]. Ahmadi, *et al*. [16] studied two-phase flow and cavity through a channel using computational analysis and investigated the effect of input Reynolds and flow properties. Aung and Yuwono [17] reported that the multi-phase flow pattern through and around the elbow bend was perhaps more dependent on liquid superficial velocity and also affected by gas volume fraction at high liquid velocity for the same operation condition. Velocity profiles and pressure at six locations on the pipe demonstrated a significant increase in elbow geometry pressure with a dramatic energy loss and loss of pressure as fluid leaves the elbow to the straight pipe section [18]. Mazumder and Siddique [19] showed that the pressure decreased at the fluid exits from the elbow and a higher pressure drop obtained at increasing air superficial velocity. Velocity and pressure drop profiles of the mixture and their cross-section contour maps by CFD simulation have been provided for distinctive multi-phase flow conducts [20]. Csizmadia and Hos [21] investigated a computational and experimental studies to determine the friction loss coefficient of power law fluids and Bingham for the flow across elbows and diffuser. In [22] were reported that the CFD contour plots will assist researchers in visualizing the fluid-flow phenomena in the pipes and bends such as boundary-layer detachment pressure and velocity profiles at high ranges of turbulent flow. Csizmadia and Till [23] studied actual losses of two non-Newtonian fluids flow in the pipes that can be defined in the power-law rheological model, the conducted study were experimental tests and computational studies. Also, they obtained the friction factor coefficient of the pipe and the loss coefficient of the short radius curvature elbow namely ($R/D = 2$). Bingham, Ostwald, Herschel-Bulkley are three non-Newtonian samples of study that are examined related to the straight pipes friction factor coefficient, the loss coefficient of the elbow and the pressure gradient that dropped on this part [24].

The key objective of this work is to examine the two-phase total pressure drop of air-water flow through 90° vertical upward to horizontal elbow and to study the effect of elbow curvature radius and pipes diameter. This work uses computational analysis in four different pipe diameters with six different curvature radius to pipe diameter ratios. The computational results were verified using experimental data, and a comparison was made between the computational results and empirical models found in the literature.

The CFD approach and geometry details

Fluid mechanics are considered as the basis in forming the CFD simulation, which is based on using algorithms and numerical approaches to analyze complicated problems such as fluid-flows. The CFD models are designed to simulate the interaction of fluids where are the surfaces determined by boundary conditions. The CFD method has become a powerful and efficient technique for understanding the intricate hydrodynamics of multi-phase flows specially gas-liquid two-phase flows and that is because of the recent advances in computer fields including the software and hardware. Several ways exist to estimate the pressure drop in 90° elbows and pipe bends, in this study the ANSYS FLUENT was employed to model the flows of gas-liquid two-phase.

Eleven 3-D 90° elbows (vertical to horizontal) were created using FLUENT design modeler to perform this analysis and to simulate pressure drop. Different curvature radius to diameter ratio, R/D, in this simulation were used ranging from (1-10) to represent the short and long radius elbows. For fully developed flow the required Le/D is approximately between (100-150) [25]. In this study, due to the difficulties and limitations of the experimental test method, the Le/D was set to be 40, as mentioned and validated by [20]. Unstructured mesh with an optimal number of nodes was used. Due to its ability to provide high quality solution, Tetrahedral mesh was created using a higher number of cells and faces than comparable hexahedral and polyhedral meshes for elbows and bends [26]. Figure 1 displays the 3-D mesh and its inlet for the elbow.

Table 1 indicates the mesh information for elbows created by ANSYS Mesh to be used for this analysis. The CFD analysis data were used in this computational study at different three locations illustrated in fig. 2.

Multi-phase modelling and assumptions

In this computational study, FLUENT solver was used. This solver is open and adjustable for a wide range of geometries. A mixture model method was used, this model is simple and reliable to apply and suitable for multi-phase flow simulation compared to other models and it is suitable for flows in pipes and bends where phases pass at different velocities by analyzing

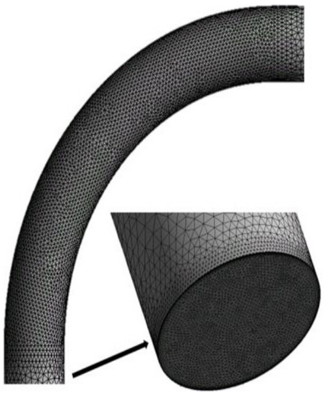

Figure 1. Elbow mesh created for simulation

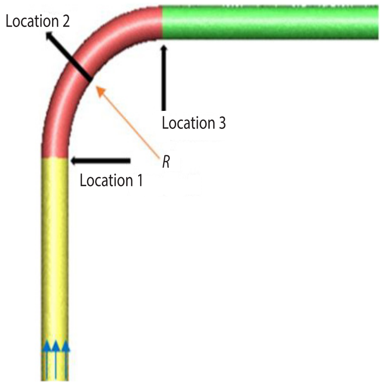

Figure 2. The 3-D elbow geometry and straight pipe sections

Table 1. Elbows details used in this study

Elbows	*D* [mm]	*R*/*D*	Number of nodes	Number of faces	Number of tetrahedral cells
Elbow 1	50	1	330829	1933542	818174
Elbow 2	50	2	404589	2378362	1008341
Elbow 3	50	4	391142	2278213	962802
Elbow 4	50	6	429221	2501660	1057409
Elbow 5	50	8	664739	3937613	1673598
Elbow 6	50	10	507886	2956169	1248867
Elbow 7	25	6	552164	3208096	1354381
Elbow 8	75	6	479752	2879527	1229318
Elbow 9	100	6	467744	2681895	1126974
Elbow 10	25	1	501377	2878082	1209809
Elbow 11	25	2	657848	2878353	1213568

continuity, momentum, and energy equations for mixtures. Also, the mixture model permits each phase to travel at various speeds, and this model apply the principle of slip velocity. While the phases travel at the same speed for the same flow conditions, the mixture model is turned to a homogeneous multi-phase model. In a vertical up flow direction, the gravitational acceleration has been taken into consideration.

In this study, the influence of changing temperature on the air-water two-phase flows was ignored and the flows were assumed to be steady-state and kept at 20 °C, where the isothermal conditions are presumed. The influence of shear stress and wall roughness on mixture flow inside the pipes and elbows was not investigated. A straight pipe section was placed before and after the bend section calculate the pressure drop around each elbow. The standard k-ε turbulence model to minimizing unknowns using equations which is suitable for a lot of turbulent flows and applications, with wall functions, was used in this analysis to solve the flow characteristics for multi-phase flow and its conditions, because it is simple and reliable model available in FLUENT. The constants of the ANSYS FLUENT model used in this study were $C_{1\varepsilon} = 1.44$, $C_{2\varepsilon} = 1.92$, $C_\mu = 0.09$, $\sigma_k = 1.0$, $\sigma_\epsilon = 1.3$. Solution methods have been adopted to increase the accuracy and convergence of the solution. When the model takes into account the slip ratio and slip velocity, a low under relaxation factors were used to obtain a uniform and stable solution steps. In order to increase the efficiency of the solver and to improve convergence, SIMPLE pressure-velocity coupling algorithm where selected, it is the default scheme and it is accurate technique to solve incompressible flows. For the momentum, the discretization scheme used was second-order upwind, whereas for the volume-fraction, the turbulent dissipation-rate and the turbulent kinetic energy first-order upwind were used. The governing equations can be solved consecutively, independent from each other, requiring less memory and running time. In mixture model, there are several differential equations to be calculated in a 3-D (k-ε) turbulence model fluid-flow process. However, there are several residuals to be monitored for accurate and convergence: continuity, 3-D velocities, k (turbulent kinetic energy), and ε (turbulent dissipation rate). The hybrid initialization is good enough for this problem to initialize the unknowns before solving the problem.

Mathematical model

The mixture models use the governing equations to solve the multi-phase fluid-flow problems, when the phases have own velocity and travel at different velocity. The mixture model is the simplest model used at various air and water velocities in case of gas-liquid two-phase flow [20].

The continuity expression of the mixture:

$$\frac{\partial}{\partial t}(\rho_\mathrm{m}) + \nabla(\rho_\mathrm{m}\overline{v}_\mathrm{m}) = 0 \tag{1}$$

where the mixture velocity, $\overline{v}_\mathrm{m}$, is:

$$\overline{v}_\mathrm{m} = \frac{\sum_{k=1}^{n} \alpha_\mathrm{k}\rho_\mathrm{k}\overline{v}_\mathrm{k}}{\rho_\mathrm{m}} \tag{2}$$

and the mixture density, ρ_m, is given:

$$\rho_m = \sum_{k=1}^{n} \alpha_k \rho_k \tag{3}$$

where α_k is the volume fraction and k – the phase.

The general momentum equation of the mixture is given by:

$$\frac{\partial}{\partial t}(\rho_m\overline{v}_\mathrm{m}) + \nabla(\rho_m\overline{v}_\mathrm{m}\overline{v}_\mathrm{m}) = -\nabla p + \nabla\left[\mu_\mathrm{m}(\nabla\overline{v}_\mathrm{m}^T)\right] + \rho_\mathrm{m}\overline{\mathrm{g}} + \overline{F} + \nabla\left(\sum_{k=1}^{n}\alpha_\mathrm{k}\rho_\mathrm{k}\overline{v}_\mathrm{dr,k}\overline{v}_\mathrm{dr,k}\right) \tag{4}$$

where $\overline{F}$ is the fluid body force in Newton, n – the phases number, and μ_m – the mixture dynamic viscosity:

$$\mu_\mathrm{m} = \sum_{k=1}^{n} \alpha_\mathrm{k}\mu_\mathrm{k} \tag{5}$$

and $\overline{v}_\mathrm{dr,k}$ is the drift velocity and k – the secondary phase

$$\overline{v}_\mathrm{dr,k} = \overline{v}_\mathrm{k} - \overline{v}_\mathrm{m} \tag{6}$$

The slip velocity or relative velocity is given:

$$\overline{v}_\mathrm{pq} = \overline{v}_\mathrm{p} - \overline{v}_\mathrm{q} \tag{7}$$

where p is the secondary phase and q – the primary phase.

The ratio of the mass (mass fraction) for phase k is given:

$$c_{\mathrm{k}} = \frac{\alpha_{\mathrm{k}}\rho_{\mathrm{k}}}{\rho_{\mathrm{m}}} \tag{8}$$

The drift velocity, $\overline{v}_{\mathrm{dr,k}}$, and the relative velocity, $\overline{v}_{\mathrm{pq}}$, are connected forming:

$$\overline{v}_{dr,p} = \overline{v}_{pq} - \sum_{k=1}^{n} c_k \overline{v}_{qk} \tag{9}$$

The relative velocity for theoretical assumption of the algorithmic slip formula of mixture model is given:

$$\overline{v}_{\mathrm{pq}} = \frac{\tau_{\mathrm{p}}}{f_{\mathrm{drag}}}\left[\frac{(\rho_{\mathrm{p}} - \rho_{\mathrm{m}})}{\rho_p}\right]\overline{a} \tag{10}$$

where τ_{p} is the particle relaxation time and given:

$$\tau_p = \frac{\rho_p d_p^2}{18\mu_q} \tag{11}$$

where d is the diameter of the secondary phase, p, in the multi-phase flow and can be (bubbles or droplets) and $\overline{a}$ is the linear acceleration traveled of the secondary phase particles. The drag function, f_{drag}, depends on the shape, size, velocity of the object, and its obtained from Schiller and Naumann [19]:

$$f_{\mathrm{drag}} = \begin{cases} 1 + 0.15\,\mathrm{Re}^{0.687}, & \mathrm{Re} \leq 1000 \\ 0.0183\,\mathrm{Re}, & \mathrm{Re} > 1000 \end{cases} \tag{12}$$

and the particle acceleration, $\overline{a}$, is given:

$$\overline{a} = \overline{\mathrm{g}} - (\overline{v}_{\mathrm{m}}\nabla)\overline{v}_{\mathrm{m}} - \left[\frac{\partial \overline{v}_{\mathrm{m}}}{\partial t} a\right] \tag{13}$$

At considering the mixture as a whole, instead of two-phases separately (drift flux method), relative velocity includes a diffusion part in turbulent flows when the time is changed, and because of the scattering in the momentum equation were occurred for the sparse phase. ANSYS Fluent applies the dispersal to the relative velocity and becomes:

$$\overline{v}_{\mathrm{pq}} = \frac{(\rho_{\mathrm{p}} - \rho_m)d_{\mathrm{p}}^2}{18\mu_{\mathrm{q}} f_{\mathrm{drag}}}\overline{a} - \frac{v_{\mathrm{m}}}{\alpha_{\mathrm{p}}\sigma_D}\nabla\alpha_{\mathrm{q}} \tag{14}$$

where v_{m} is the mixture kinematic viscosity and σ_D – the Prandtl dispersal coefficient.

Empirical models and experimental investigation

Several empirical models in experimental test systems have been studied to calculate the total pressure drop in elbows for multi-phase flows. The models apply same theories because of no reference to the particular flow pattern provided in the pipe bend and elbow. These theories were conducted and improved using and applying experimental data presented and developed by a number of authors [27]. In order to support and validate this computational study and to find similar results that validate this CFD study, pressure drop calculations for pipes and bends were performed using Azzi and Friedel model [28], Chisholm [9], and Benbella *et al.* [2] empirical models conducted on elbows for similar boundary and flow conditions. Chisholm proposed a model, that includes dimensionless parameters achieved by experimental data for two-phase flow, and prepared equation that is a combination of the homogeneous volumetric form and the Dean number [29]. A multi-phase air-water system was built and managed to validate computational study (CFD simulation) and empirical models results. The schematic lay-out for the system loop can be seen in fig. 3. The system includes a control input supply system, a water tank with required pipe-lines and an air system with also pipe-lines and its gauges. The water was pumped and circulated in the system through a centrifugal pump with a range of (0.05-5 kg/s) and a rotameter was used to calculate the water flow rate up to 21 m^3 per hour. In the meantime, the air was supplied in the loop by an air compressor with the air-flow rate to count the required air supply its range is (0.00006-3.5 kg/s) and with a flow regulator with a range of (0-30 m^3 per hour). A mixing chamber (mixer) was also used to distribute the air in the water stream. In addition the water and air-flow meters used as basic instruments, the pressure gauge and the differential pressure transducers (Comark C9500 Series) were installed to determine the pressure difference in each point through the parts (the calibrated precision of ±0.1% within the range). The test section consists of (25 mm) inner-diameter, *D*, PVC straight pipe segments connected with the elbow to form the total geometry, 90° elbow with vertical and horizontal sections. These sections are necessary for obtaining fully developed flow and for well mixed mixture, in order to increase the accuracy and efficiency of the system. The installed lengths of the vertical and horizontal parts nearly identified in length and they equal to 1.0 m and 0.95 m, with development lengths of 40*D* and 38*D*, respectively.

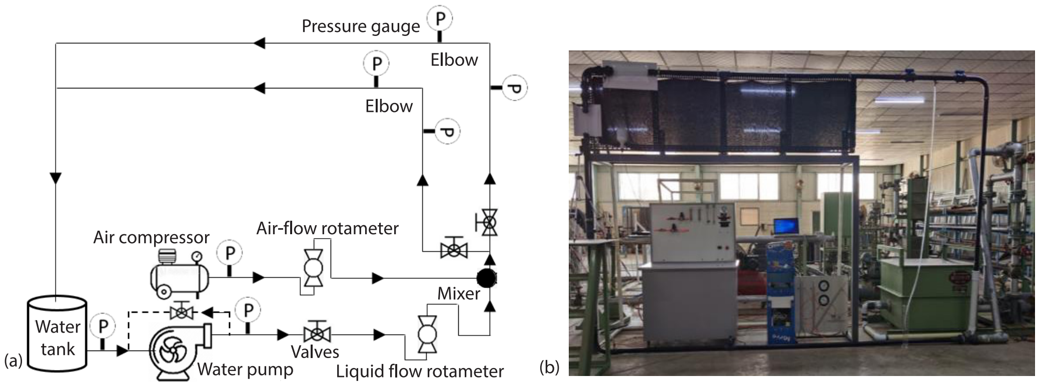

Figure 3. Experimental rig; (a) schematic diagram and (b) photo of two-phase flow laboratory

The CFD analysis and results

Computational analysis was conducted on eleven different elbows with six different curvature radius to diameter ratios, at mass qualities (0.01, 0.1, 0.2, 0.3, 0.4, 0.5) and mass flux (350, 450, 550, 750, 1000 kg/m^2s) for pipe diameter (25, 50, 75, 100 mm). Each of these boundary and flow conditions were analyzed in CFD to compute the influence of varying pipe diameters and *R*/*D* ratios on pressure drop for each individual elbow. The experimental test system has several difficulties and limitations; it was not possible to carry out the experimental investigation for all these conditions.

Validation of CFD results

Figure 4 demonstrates a comparison between computational, experimental, and predicted results for air-water two-phase flow total pressure drop in a vertical upward 90^o elbow bend and at different water velocities. The predicted results were taken from correlation available in literature, Azzi and Friedel [28], Chisholm [9], and Benbella *et al*. [2]. The experimental, the correlations and the CFD results showed a similar trend and behavior for the same flow conditions with slight difference in values, the pressure differences and the total pressure drop increases with raising both water and air velocities. Whereas, pressure drop decreases significantly with increasing curvature radius to diameter ratio of the elbow. Due to increasing in momentum with decreasing curvature radius, R, therefore, the pressure drop was duplicated with doubling the curvature radius to diameter ratio, R/D, with remaining the other parameter constant (*i.e.* pipe diameter, air velocity, water velocity), as illustrated in figs. 4(a) and 4(c) and figs. 4(b) and 4(d).

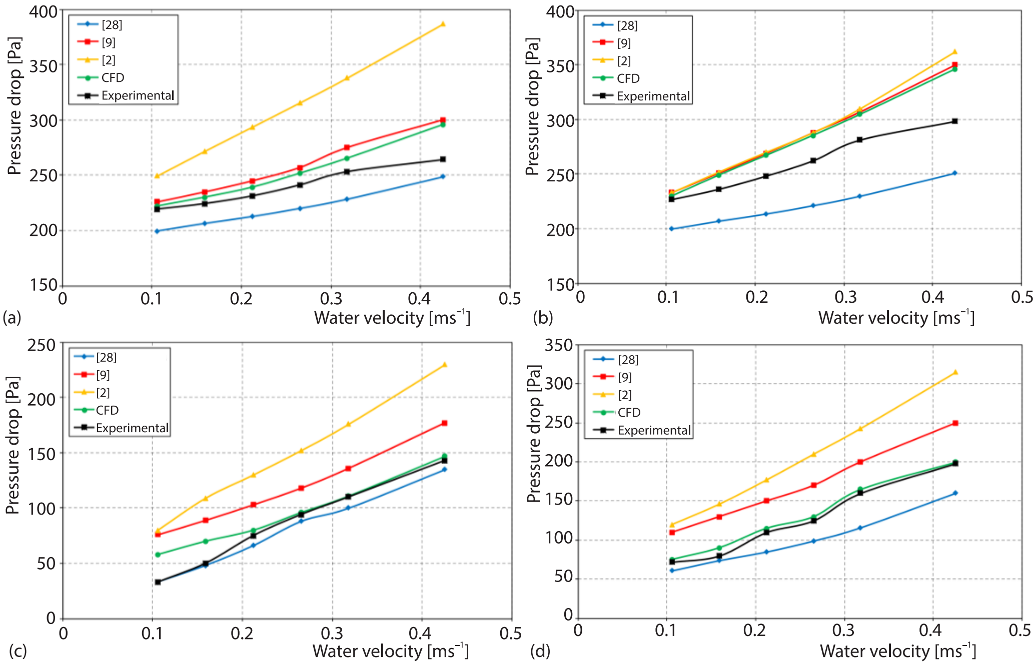

Figure 4. Comparison between experimental, computational and different correlations for pressure drop *vs.* water velocity D = 25 mm; (a) R/D = 1 and V_a = 0.212 m/s, (b) R/D = 1 and V_a = 0.425/s, (c) R/D = 2 and V_a = 0.212 m/s, and (d) R/D = 2 and V_a = 0.425 m/s

Effect of mass flux and mass quality

The pressure drops due to flow resistance as a fluid-flow inside the pipe and bends, this pressure loss depends on many factors and parameters when the fluid-flows through the pipe and also, depends on the pipe geometries and the orientations. The pressure drop strongly depends on Reynolds and Froude numbers and mass quality [30]. The computational results performed by ANSYS FLUENT of the total pressure loss of the two-phase flow in vertical upward 90° elbow as a function of mass quality and mass flux are illustrated in fig. 5. For these flow conditions and boundaries, and to investigate the direct effect of mass flux and mass quality, the pipe diameter and curvature radius to diameter ratio have been fixed and they are equal to D = 50 mm and R/D = 4. It is obvious from the fig. 5 that the pressure drop of the mixture increased significantly with increasing mass quality and mass flux, it can be noted that the pressure drop increased from 2-450 kPa, when the mass flux and mass quality are high and equal to 750 kg/m^{2}s and 0.5, respectively. However, the maximum predicted pressure dropped at a high range of mass flux and mass quality, because in these range the air velocity is higher and flows in the core of the pipe and the water flow near the pipe walls resulting maximum possible of friction. Moreover, variation of mass flux and mass quality caused by changing fluid velocity, Water hammer is a temporary fluid occurrence caused by a sudden change in fluid velocity in hydraulic systems. In pressurized pipe-line systems, the sudden shutdown of a pump or the closure of a valve causes fluid transients that can produce extremely high pressures. During transient events that result in column separation, very low pressures may also occur [31].

The curvature radius to diameter ratio is a parameter plays a role in design of desired shape and size of elbows and pipe bends in engineering applications. Also, the curvature radius which is the radius measured to the center line of the pipe has a significant effect on some of undesired phenomena, such as back pressure, centrifugal force, phases dispersion, mixture resistance*, etc.*

The curvature radius has a significant effect on the flow behaviors at the elbow's outlet. Therefore, in this study various curvature radius to diameter ratio has been investigated to reveal and to detect the influences these ratios on swirling intensity distribution and turbulent intensity. Increasing curvature radius leads to make the flow turbulent intensity after elbow's outlet smoother. Also, the curvature radius has an effect on swirling intensity distribution, increasing the elbow radius leads to increasing in the swirling intensity after elbow's outlet. In other words, the flow field tends to form a more uniform turbulence distribution after passing through the elbow, whereas the intensity of the turbulence tends to allocate beside of the outer wall [32].

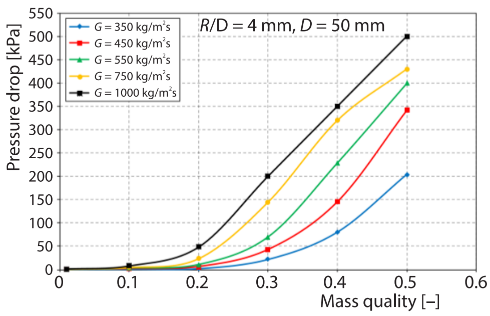

Figure 5. Two-phase flow predicted pressure drop *vs.* mass quality for different mass flux

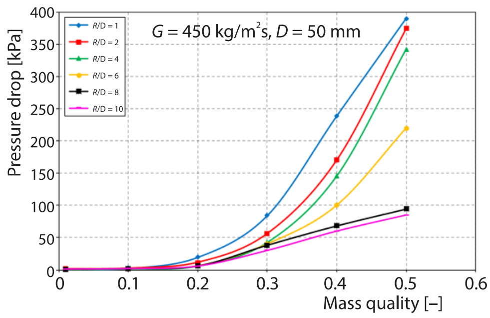

Figure 6. Two-phase flow predicted pressure drop *vs.* mass quality for different curvature ratios

To investigate the influence of bend curvature radius in this study, the curvature radius to diameter ratio was varied methodically, from 1 which represent the small bend radius up to 10 that represents the large bend radius. The total pressure drop *vs*. the curvature radius to diameter ratio and the mass quality is demonstrated in fig. 6. Hence, from the fig. 6 the pressure drop is higher at low curvature radius to diameter ratios at a constant mass flux of 450 kg/m^{2}s and for pipe diameter of 50 mm. The pressure drop is increased from 2 to near 300 kPa with decreasing curvature radius to diameter ratio gradually from 10 to 1. In the other words, a smaller curvature radius to diameter ratio results in a higher pressure drop and a weak developing mixture flow at the bend portion. On the other hand, a larger curvature radius to diameter ratio results in a lower pressure drop and a strong developing two-phase flow at the bend section.

Influence of pipe diameter

The geometry parameters have a dramatic influence on the total pressure drop of the two-phase flow in pipes and bends. In this computational investigation, the pipe diameter is one of the paramount importance and effective parameter. Figure 7 illustrates the effect of the inner diameter of the pipe on the total pressure drop at the different mass quality and at the same mass flux and curvature radius to diameter ratio for computational and empirical results. Also, the total pressure drop is higher when the mass quality is high and the total pressure loss rises slightly with increasing pipe diameters. Although, the pipe diameter has a slight effect on two-phase pressure drop. The overall decrease in fluid pressure is due to friction, mixture velocity, potential energy and geometry parameters (curvature radius). In general, the two-phase pressure drop in pipes and fittings is due to a change in kinetic energy and potential energy (gravity effect). Also, due to resistance of the pipe wall and its interaction with the fluid-flow (friction effect) and due to geometry of the pipes and bends such as length, diameter, bend curvature radius, wall surface roughness, and, *etc.*

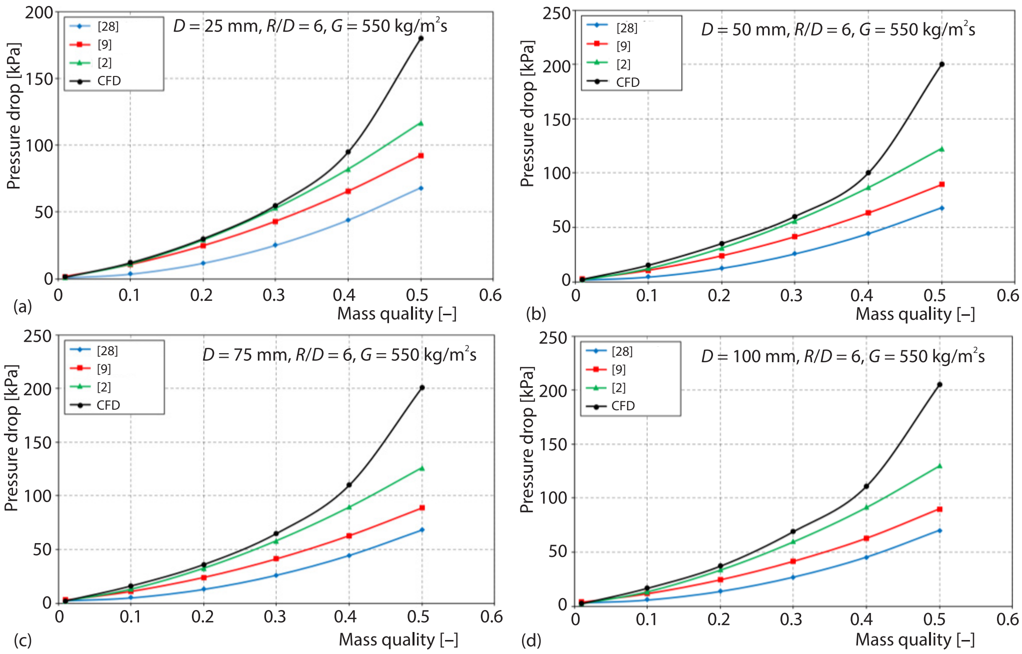

Figure 7. Comparison between the present computational results and different predicted results for pipe diameter; (a) *D* = 5 mm, (b) *D* = 50 mm, (c) *D* = 75 mm, and (d) *D* = 100 mm

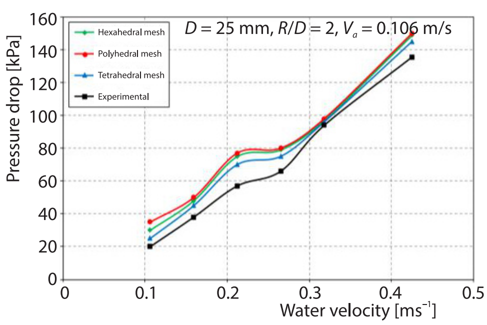

Figure 8. Experimental and computational results for three different mesh types

Effect of CFD meshing

The mesh affects the quality, convergence and speed of the simulation. Furthermore, since CFD meshing usually takes a considerable some of the time to obtain desirable simulation results, the stronger and more efficient the meshing methods, the quicker and more reliable the solution. Cleary this analysis demonstrates that the tetrahedral mesh keeps a great promise in producing adequate accuracy results as a comparison other mesh types with the added benefits of getting converge, but with higher iterations and lower solution runtimes. Also, robust convergence to lower residual values, as shown in fig. 8 the tetrahedral mesh is more converge to the experimental data for the same pipe diameter, curvature radius to diameter ratio and air velocity compared to hexahedral and polyhedral meshes. Also, the tetrahedron cells are suitable for all kinds of pipe geometries simple and complicated geometry taking the mesh quality into account, good mesh quality the accurate solution, CFD mesh is capable of examining and check the simulation mesh quality by checking the aspect ratio, skewness, element quality and*, etc.*

The CFD output (contours and plots)

The design of an effective elbow bend is difficult in the fluid separation and mixing processes. As long as the elbow bends become desirable in many engineering applications, the phase distribution process (flow pattern) is important and pressure control is characteristic of air-water two-phase for 90^o elbow bend with two sections of straight pipe is analyzed theoretically, computationally and experimentally. Figure 9 shows absolute pressure contour plots for

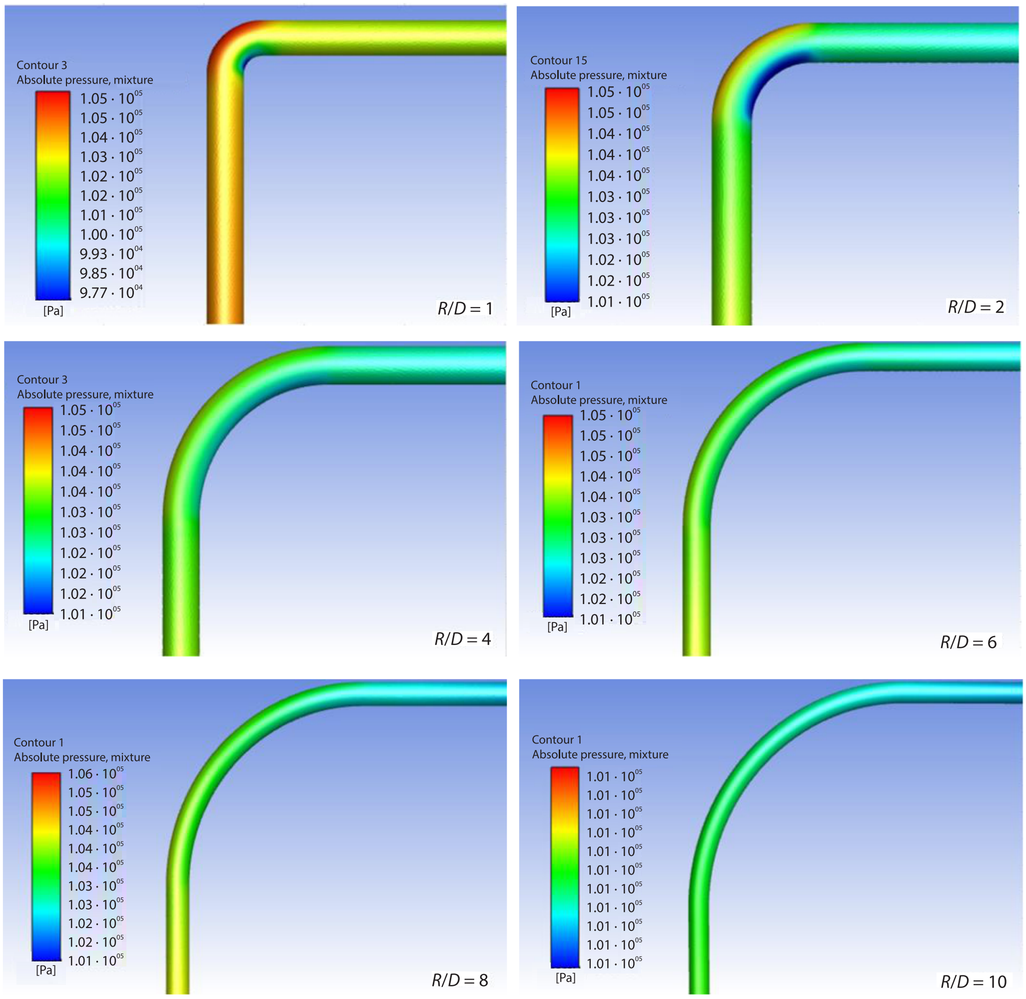

Figure 9. Effect of R/D on pressure drop at V_a = 30 m/s and V_w = 0.5 m/s and for 50 mm pipe diameter

pipe diameter 50 mm with six different curvature radius to diameter ratios 1, 2, 4, 6, 8, and 10, with air and water velocities 30, 0.5 m/s, respectively. It can be noticed from fig. 9 that the pressure distribution at the elbows indicates irregular pressure distribution. Also, from the front view of the pressure counters, the pressure magnitude is low near the inner wall of the elbows. Whereas, higher pressure can be seen from the outer wall of the elbows. Also, higher absolute pressure can be noticed in the inlet section pipe before elbows, while lower absolute pressure can be shown in the outer section pipe after elbows. The mixture pressure is dramatically higher at the outer surface of the elbow for a low curvature radius to diameter ratio as indicated in fig. 9. Hence, a higher R/D ratio resulted in a lower pressure drop.

Cross-sectional radial velocity and absolute pressure contours are presented in fig. 10. Location 1 from fig. 2 is the outlet of the up flow pipe portion (end of the vertical section) and it is the entrance of the elbow section, while Location 3 from the same figure is the outlet of the elbow section and the entrance of the down flow pipe portion (an inlet of the horizontal section), and Location 2 is the cross-section (mid- center) of the elbow. From fig. 10 the inner and outer walls of the elbows are bottom and top of each cross-section contours, respectively. The velocity intensity decrease can be shown at Location 3 at $R/D = 1$, while at $R/D = 2$, the velocity intensity decreasing relatively less. The pressure distribution from the contours is scattered without showing any uniform pattern, but it can be noticed that the pressure scale decreased at Location 3 because of the bend effect and decreased with increasing the curvature radius to diameter ratio, while the pressure raised with increasing the air velocity.

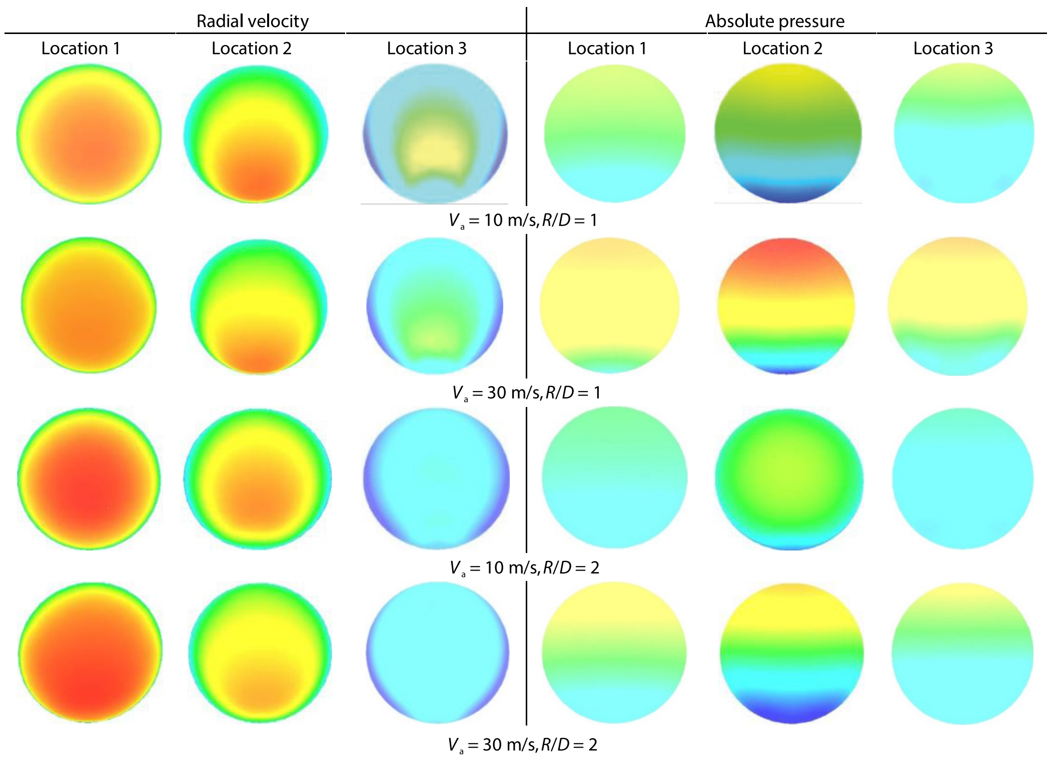

Figure 10. Velocity and absolute pressure at three different elbow locations for V_w = 0.5 m/s and D = 50 mm

Conclusions

The CFD analysis with ANSYS 19.2 FLUENT, experimental study and empirical models were performed for air-water two-phase flow in a vertical to horizontal 90° elbow with four different pipe diameters (25, 50, 75, and 100 mm) with R/D ratios of (1, 2, 4, 6, 8, and 10). Analysis was performed for different air velocities, different water velocities, mass fluxes (350, 450, 550, 750, and 1000 kg/m^{2}s) and for mass qualities (0.01, 0.2, 0.3, 0.4, 0.5). The mixture model is also used to account for various velocities of air and water as well as mass quality to solve governing equations (continuity, momentum, and energy equations). Pressure drop data and mixture velocity with their associated cross-section contour maps were already provided for characteristic multi-phase flow behaviors.

The CFD analysis revealed a decrease in total pressure as the mixture exits the elbow to the straight pipe segment. Higher pressure drops at higher air velocity at constant water velocity were observed. Also, pressure drops increased at both high mass flux and mass quality. The increase of mass quality also caused a turbulent gas pattern after the elbow bend. Slight pressure loss was observed when pipe inner diameter increased from (25-50 mm, 75 mm and 100 mm), whereas a larger drop was observed when curvature radius to diameter ratio decreased. The investigated parameters such as low R/D, high mass flux and mass quality increased the total pressure drop due to the increase of the effect of interaction between two-phases, centrifugal forces formation, separation of two-phases or slip and friction between the air and the water. CFD meshing analysis showed that tetrahedral cells produced equivalent accuracy results compared to hexahedral and polyhedral meshes with the added benefits of convergence, but with higher iterations and lower solution runtimes. The results of the CFD were validated with the experimental data and compared with three separate correlations available in the literature, and a good agreement was achieved.

Nomenclature

a – acceleration, [ms^{-2}]
D – inner pipe diameter, [mm]
f – friction factor for pipe
F – force, [N]
G – mass flux, [kgm^{-2}s^{-1}]
g – gravitational acceleration, [ms^{-2}]
k – friction factor for bend
Le – equivalent pipe length, [mm]
R – elbow curvature radius, [mm]
Δp – two-phase pressure drop, [kPa]
R/D – elbow curvature radius to pipe diameter ratio
Re – Reynolds number
t – time
v – velocity, [ms^{-1}]
V_a – air velocity, [ms^{-1}]
V_w – water velocity, [ms^{-1}]
x – dryness fraction/mass quality

Greek symbols

α – volume fraction
μ – dynamic viscosity, [kgm^{-1}s^{-1}]
ρ – density, [kgm^{-3}]
σ – Prandtl coefficient
τ – relaxation time, [s]

Subscripts

dr – drift
drag – drag force
k – any phase
m – mixture
p – secondary phase
pq – relative phase
q – primary phase

References

[1] Spedding, P. L., *et al*., Fluid-flow Through 90° Bends, *Developments in Chemical Engineering and Mineral Processing*, *12* (2004), 1-2, pp. 107-128
[2] Benbella, S., *et al*., Gas-Liquid Pressure Drop in Vertical Internally Wavy 90-Degree Bend, *Experimental Thermal and Fluid Science*, *33* (2009), 2, pp. 340-347
[3] Wallis, G. B., *The 1-D Two-Phase Flow*, McGraw-Hill, New York, USA, 1969
[4] Cole, J. S., *et al*., Friction Factors in Two-phase Horizontal Pipe Flow, *International Communications in Heat and Mass Transfer*, *31* (2004), 7, pp. 909-917

[5] Wongwises, S., Kongkiatwanitch, W., Interfacial Friction Factor in Vertical Upward Gas-Liquid Annular Two-Phase Flow, *International Communications in Heat and Mass Transfer*, *28* (2001), 3, pp. 323-336
[6] Spedding, P. L., *et al*., Prediction of Pressure Drop in Multi-phase Horizontal Pipe Flow, *International Communications in Heat and Mass Transfer*, *33* (2006), 9, pp. 1053-1062
[7] Chenoweth, J. M., Martin, M. W., Turbulent Two-Phase Flow, *Pet. Ref*., *34* (1955), 10, pp. 151-155
[8] Lockhart, R. W., Martinelli, R. C., Proposed Correlation of Data for Isothermal Two-Phase Two-Component Flow in Pipes, *Chem. Eng. Prog*., *45* (1945), 1, pp. 39-48
[9] Chisholm, D., *Two-Phase Flow in Pipe-lines and Heat Exchangers*, Godwin, N. C., USA, 1983
[10] Arun, A., *et al*., An Experimental Study on Two-Phase Pressure Drop in Small Diameter Horizontal, Downward Inclined and Vertical Tubes, *Thermal Science*, *19* (2015), 5, pp. 1791-1804
[11] Sekoda, K., *et al*., Horizontal Two-Phase Air Water Flow Characteristics in the Disturbed Region Due to a 90-Degree Bend, *Japan Society Mechanical Engineering*, *35* (1969), 289, pp. 2227-2333
[12] Deobold, T. L., An Experimental Investigation of Two-Phase Pressure Losses in Pipe Elbows, M. Sc. thesis, University of Idaho, Chemical Engineering, Moscow, Id., USA, 1962
[13] Alves, G. E., Co-Current Liquid-Gas-flow in a Pipe-Line Contactor, *Chemical Engineering Progress*, *50* (1954), 9, pp. 449-456
[14] Peshkin, M. A., About The Hydraulic Resistance of Pipe Bends to the Flow of Gas-Liquid Mixtures, *Teploenergetika*, *8* (1961), 6, pp. 79-80
[15] Kutateladze, S. S., *Problems of Heat Transfer and Hydraulics of Two-Phase Media*, Pergamon Press, Oxford, UK, 1969
[16] Ahmadi, M., *et al*., Computational Fluid Dynamic Simulation of Non-Newtonian Two-Phase Fluid-flow through a Channel with a Cavity, *Thermal Science*, *24* (2020), 2B, pp. 1045-1054
[17] Aung, N. Z., Yuwono, T., Computational Fluid Dynamics Simulations of Gas-liquid Two-phase Flow Characteristics through a Vertical to Horizontal Right Angled Elbow, *ASEAN Journal on Science and Technology for Development*, *30* (2013), 1-2, pp. 1-16
[18] Mazumder, Q. H., The CFD Analysis of Single and Multi-phase Flow Characteristics in Elbow, Mechanical Engineering, University of Michigan-Flint, Flint, USA, *Engineering*, *4* (2012), 4, pp. 210-214
[19] Shiller, L., Nauman, A. Z., A Drag Coefficient Correlation, *Verein Deutschen Ingenieure Zeitung*, *77* (1993), pp. 318-320
[20] Mazumder, Q. H., *The CFD Analysis of the Effect of Elbow Radius on Pressure Drop in Multi-Phase Flow*, Mechanical Eng. Univ. of Michigan-Flint, Flint, Mich., USA, 2012, ID125405, p. 8
[21] Csizmadia, P., Hos, C., The CFD-Based Estimation and Experiments on the Loss Coefficient for Bingham and Power-Law Fluids through Diffusers and Elbows, *Comp. Fluids*, *99* (2014), July, pp. 116-123
[22] Perumal, K., Ganesan, R., The CFD Modelling for the Estimation of Pressure Loss Coefficients of Pipe Fittings: An Undergraduate Project, *Computer Appl. in Eng. Education*, *24* (2015), 2, pp. 180-185
[23] Csizmadia, P., Till, S., Experimental and Numerical Investigation of the Loss Coefficient of a 90° Pipe Bend for Power-Law Fluid, *Periodica Polytechnica Chemical Engineering*, *64* (2020), 4, pp. 469-478
[24] Csizmadia, P., Till, S., The Effect of Rheology Model of an Activated Sludge on the Predicted Losses by an Elbow, *Periodica Polytechnica Mechanical Engineering*, *64* (2018), 4, pp. 305-311
[25] Mazumder, Q. H., *et al*., Prediction of Solid Particle Erosive Wear of Elbows in Multi-phase Annular Flow-Model Development and Experimental Validation, *Journal of Energy Resources Technology*, *130* (2008), 2, ID023001
[26] ***, ANSYS Fluent Tutorial Guide, ANSYS, Inc. Release 18.0, ANSYS Europe, Southpointe, 2017
[27] Hetsroni, G., Ed., *Handbook of Multi-Phase Systems*, Hemisphere McGraw Hill, New York, 1982
[28] Azzi, A., Friedel, L., Two-Phase Upward flow 90◦ Bend Pressure Loss Model, *Forschung im Ingenieurwesen*, *69* (2005), 2, pp. 120-130
[29] Sanchez, S. F., *et al.*, Pressure Drop Models Evaluation for Two-Phase Flow in 90 Degree Horizontal Elbows, *Ingenieria Mecanica Techilogia y Desarrollo*, *3* (2010), 4, pp. 115-122
[30] Autee, A., *et al*., Experimental Study on Two-Phase Pressure Drop of Air-Water in Small Diameter Tubes at Horizontal Orientation, *Thermal Science*, *18* (2014), 2, pp. 521-532
[31] Warda, H. A., *et al*., Computational Fluid Dynamics (CFD) Simulation of Liquid Column Separation in Pipe Transients, *Alexandria Engineering Journal*, *59* (2020), 5. pp. 3451-3462
[32] Zahedi, R., Babaee Rad, A., Numerical and Experimental Simulation of Gas-Liquid Two-Phase Flow in 90-Degree Elbow, *Alexandria Engineering Journal*, *61* (2022), 3, pp. 2536-2550

Paper submitted: March 22, 2021
Paper revised: October 20, 2021
Paper accepted: November 9, 2021