

ASIEGBU, N.M., HOSSAIN, M., DROUBI, G.M. and ISLAM, S.Z. 2023. Investigation of the effects of pipe diameter of internal multiphase flow on pipe elbow vibration and resonance. *Proceedings of the Institution of Mechanical Engineers, part E: journal of process mechanical engineering* [online], 237(4), pages 1319-1330. Available from: <https://doi.org/10.1177/09544089221115520>

Investigation of the effects of pipe diameter of internal multiphase flow on pipe elbow vibration and resonance.

ASIEGBU, N.M., HOSSAIN, M., DROUBI, G.M. and ISLAM, S.Z.

2023

Users who access an article in a repository may use the article in any manner consistent with the terms of the Creative Commons license attached to the article. For more information on the terms of use and summary of the Creative Commons licenses, please see our [Reusing Open Access and Sage Choice Content](#) webpage.

Investigation of the effects of pipe diameter of internal multiphase flow on pipe elbow vibration and resonance

Nkemjika Mirian Asiegbu , Mamdud Hossain ,
Ghazi Mohamad Droubi and Sheikh Zahidul Islam

Proc IMechE Part E:

J Process Mechanical Engineering

2023, Vol. 237(4) 1319–1330

© IMechE 2022



Article reuse guidelines:

sagepub.com/journals-permissions

DOI: 10.1177/09544089221115520

journals.sagepub.com/home/pie



Abstract

Computational fluid dynamics modelling of internal two-phase flow induced transient forces at 90° elbows have been carried out to evaluate the effect of pipe diameter on the characteristics of multiphase flow induced vibration. Simulations of two-phase flows of slug, cap bubbly and churn induced vibration at a pipe elbow were carried out using the volume of fluid model for the two-phase flows and the $k - \epsilon$ model for turbulence. Modal analysis has been carried out to evaluate the risk of resonance. Results were compared across three geometrically similar pipes of different diameters. Simulation results showed that the behaviours of the flow induced forces at the pipe elbow as a function of gas velocity for internal diameters of 0.0525 and 0.2032 m are similar. However, the multiphase flow induced force characteristics are different in the 0.1016 m diameter (intermediate) pipe. It can be attributed to the transition behaviour of gas–liquid two-phase flows caused by Taylor instability in an intermediate sized pipe. The predicted root-mean-square flow induced forces as a function of Weber number were correlated with an existing empirical correlation for a wider range of pipe sizes and gas volume fractions between 40% and 80%. Furthermore, the pipe natural frequencies increase with the increase of gas volume fraction in smaller pipes and the resonance risk increases with the increase of pipe diameter.

Keywords

Two-phase flow induced vibration, numerical modelling, force fluctuations, natural frequency, resonance

Date received: 29 December 2021; accepted: 20 June 2022

Introduction

Internal multiphase flow induced vibration (MFIV) in pipe elbows poses significant fatigue failure risks in the piping network of oil and gas production, nuclear power plant and chemical process systems.^{1,2} Current industry guideline has been developed considering single-phase flows, while the few existing MFIV models reported in literature are based on small-scale laboratory experiments which do not completely address the differing multiphase flow mechanisms between small and large pipes. In addition, literature shows that MFIV is two-phase flow regime dependent with the slug and churn flows having the most significant forcing functions.^{1,3} Since flow regimes behaviours (especially slug and churn flow) and transition differ in small and large diameter pipes, the force correlations developed from experiments and modelling from small pipes can lead to costly flow assurance, pipeline integrity and safety problem if applied to large diameter pipes. Kataoka and Ishii⁴ defined the critical diameter at which large slug bubble could no longer be sustained in a pipe as:

$$D_H^* = \frac{D_H}{\sqrt{\frac{\sigma}{g\Delta\rho}}} \geq 40 \quad (1)$$

where D_H , σ , g and $\Delta\rho$ are the hydraulic diameter, surface tension, gravity and density difference between gas and liquid. Pipes having $D_H^* \geq 40$ are termed large pipes in which cap bubbles rather than long slugs exist, and $D_H^* < 18.6$ are termed small pipes while $18.6 < D_H^* \leq 40$ represent intermediate size where transition behaviour involving the disintegration of long slugs into cap bubbles is present. Therefore, developing a database of dominant frequencies, magnitudes and RMS of force fluctuations due to two-phase flows in different sized pipes will be valuable to engineers at design stage of multiphase flow systems.

Experimental studies on MFIV in large pipes are few. The most relevant of such studies were conducted by Nennie and Belfroid⁵ and Belfroid et al.⁶ These experiments were also used to validate computational fluid

School of Engineering, Robert Gordon University, Aberdeen, UK

Corresponding author:

Nkemjika Mirian Asiegbu, School of Engineering, Robert Gordon University, Sir Ian Wood Building, Garthdee Road, Aberdeen, AB10 7GJ, UK.

Email: nkemjika.asiegbu@gmail.com

dynamics (CFD) studies that were conducted in a subsequent phase of the project.^{7–9} These studies concluded that the fluctuations of liquid hold-up resemble the multiphase flow induced force fluctuations at pipe elbows. Belfroid et al.¹⁰ summarised the results of all the test cases that were investigated by Nennie and Belfroid.⁵ Their findings on the relationship between the non-dimensional force fluctuation and Weber number compared fairly to the model by Riverin et al.¹¹ and data from Liu et al.³

Most of the reported literature on FIV in pipe bends in large pipes are for nuclear power plant applications.¹² Yamano et al.¹³ results prove that both pipe size and configuration have effects on MFIV characteristics. The cost and hazard of comprehensive life scale experiments of oil and gas and nuclear power plant pipework could be prohibitive. Hence recent FIV investigations were mainly focused on numerical methods.^{14–16} Pontaza et al.⁹ applied the volume of fluid (VOF) model for two-phase flows and large eddy simulation (LES) approach in their CFD investigation of MFIV and concluded that the standard deviation of flow induced forces agreed with the experimental measurements within $\pm 20\%$. A similar study was carried out by Emmerson et al.⁷ using the CFD approach and their results showed that CFD was able to predict the magnitude of force fluctuation within 2–7%. Studies have applied U-RANS approach to capture the mechanism of FIV in large pipe bends due to pressure fluctuations, dominant in single-phase flows.^{12,13,17} In general, the risk of FIV increases in multiphase flows.¹⁸

Additionally, a complete assessment of the dynamic behaviour of a structure vibrating under loading includes the determination of its modal parameters. Kim and Srinil¹⁹ investigated a full-scale M-shaped subsea jumper transporting slug flows and established the resonance risk because the predominant frequencies of the stress and displacement fluctuations were similar to the natural frequencies of the jumper. Finally, Wang et al.²⁰ observed in their numerical study that the first three orders of natural frequencies of a pipeline conveying gas-liquid two-phase flow, increased as the gas volume fraction at pipe inlet increased. Their findings also showed that high amplitude and frequency of pressure and displacement fluctuations observed within the slug flow regime could induce excessive stress and fatigue failure in the pipeline.

In addition, the effect of added mass due to internal multiphase flow is crucial in evaluating the natural frequencies of a pipe structure and should not be ignored in an MFIV assessment. However, this effect has not been quantified and compared for different pipe sizes so far. Therefore, the present study has implemented a validated CFD approach²¹ to model slug/cap-bubbly to churn-turbulent flow regime induced force fluctuations in pipes of 0.1016 m I.D. (intermediate size) and 0.2032 m I.D. (large size). A comparison of key FIV parameters of dominant frequency and RMS of flow induced force fluctuations has been carried out between previously

reported laboratory scale pipe diameter of 0.0525 m I.D. (small) and the newly obtained data from intermediate and large pipes. Moreover, the effect of added mass due to internal multiphase flow on the natural frequencies of a pipe structure as a function of pipe diameter has been studied.

Numerical methods

Volume of fluid method

This study implements the VOF method to model the flow of air and water by solving a single set of momentum equations. In this method, an equation representing the conservation of the volume fraction α and one continuity equation for both phases are also solved. The VOF method tracks the volume fraction of each phase through the computational domain.^{22,23} The volume fraction continuity equation of one phase given in equation (2) is used to track the phases:

$$\frac{\partial \alpha_q}{\partial t} + \bar{u} \cdot \nabla \alpha_q = 0 \quad (2)$$

where subscript q represents each phase component, $\bar{u}$ is velocity vector, t is time and α represents the volume fraction of one of the phases.

The volume fraction of the other phase is obtained from the relation

$$\sum_{q=1}^2 \alpha_q = 1 \quad (3)$$

The continuity and momentum equations are solved alongside the volume fraction equation. They are given as, continuity:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \bar{u}) = 0 \quad (4)$$

Momentum:

$$\frac{\partial (\rho \bar{u})}{\partial t} + \nabla \cdot (\rho \bar{u} \bar{u}) = -\nabla P + \nabla \cdot \mu (\nabla \bar{u} + \nabla \bar{u}^T) + F$$

where ρ , μ and F are density, dynamic viscosity and external body forces, respectively. The properties appearing in the transport equation are calculated by summing the contributions of all phases in the multiphase flow according to the volume fractions of the phases. For example, equation (6) accounts for the contribution of each phase to the density of the two-phase flow.

$$\rho_m = \sum_{q=1}^2 \alpha_q \rho_q \quad (6)$$

ρ_m is mixture density. The term F in the momentum equation is used to account for the surface tension force between liquid water and air. F is given as:

$$F = \sigma \left(\frac{\rho k_1 \nabla s_1}{1 / 2(\rho_l + \rho_g)} \right) \quad (7)$$

Where, σ is the surface tension coefficient, and κ_1 is the surface curvature of the liquid droplet defined in terms of the divergence of the unit normal $\underline{n}_1$. κ_1 is given by

$$\kappa_1 = \nabla \cdot \underline{n}_1 \quad (8)$$

The local gradients in the surface normal to the interface are employed to account for n_1 :

$$\underline{n}_1 = \frac{\nabla s_1}{|\nabla s_1|} \quad (9)$$

s represents surface and $\underline{n}_1$ is unit normal.

Turbulence model

The mixture $k - \epsilon$ turbulence model has been implemented in the present study. Launder and Spalding²⁴ gave the mixture turbulent kinetic energy k and energy dissipation rate ϵ as equations (10) and (11), respectively:

$$\nabla \cdot (\rho_m \vec{v}_m k) = \nabla \cdot \left(\frac{\mu_{t,m}}{\sigma_k} \nabla k \right) + G_{k,m} - \rho_m \epsilon \quad (10)$$

$$\nabla \cdot (\rho_m \vec{v}_m \epsilon) = \nabla \cdot \left(\frac{\mu_{t,m}}{\sigma_\epsilon} \nabla \epsilon \right) + \frac{\epsilon}{k} (C_{1\epsilon} G_{k,m} - C_{2\epsilon} \rho_m \epsilon) \quad (11)$$

the mixture density and velocities are given by:

$$\rho_m = \sum_{i=1}^N \alpha_i \rho_i \quad (12)$$

and

$$\vec{v}_m = \frac{\sum_{i=1}^N \alpha_i \rho_i \vec{v}_i}{\sum_{i=1}^N \alpha_i \rho_i} \quad (13)$$

$\mu_{t,m}$ is the turbulent viscosity and is calculated from:

$$\mu_{t,m} = \rho_m C_\mu \frac{k^2}{\epsilon} \quad (14)$$

and the production of turbulent kinetic energy, $G_{k,m}$ is computed from:

$$G_{k,m} = \mu_{t,m} (\nabla \underline{v}_m + (\nabla \underline{v}_m)^T) : \nabla \underline{v}_m \quad (15)$$

The turbulent model constants are:

$$C_\mu = 0.09,$$

$$C_{1\epsilon} = 1.44, C_{2\epsilon} = 1.92, \sigma_k = 1.0, \sigma_\epsilon = 1.3$$

The standard $k-\epsilon$ URANS turbulence model requires a modest computing resource to produce a reasonably accurate solution compared to the rest models and approaches of calculating turbulence. It is well established and performs within reasonable accuracy for many industrial flows. The unsteady $k-\epsilon$ model cannot predict the turbulence pressure fluctuations, however, unlike single-phase flow, the effect of the impact forces on bends due to density difference of the alternating slugs and large interface waves, is more significant than the effect of turbulence fluctuations in two-phase slug/churn FIV.

Computational geometry

Three different pipe sizes presented in Table 1 were studied, the pipe sizes were categorised according to the critical diameter criteria. The pipe geometry is shown in Figure 1. Base case pipe geometry represented in Table 1 as small is taken from Liu et al.³

Inlet and outlet boundary conditions

Based on the entrance length mechanism suggested by Taitel et al.²⁵ for slug-churn flow regime, with sufficiently long entrance length, multiphase flows introduced into a flow channel at individual phase velocities will eventually separate into the flow patterns corresponding to the superficial velocities. However, the concentric two inlets method described in Parsi et al.²⁶ has been utilised in the present study, where air and water are introduced into the pipe through two separate inlets. This method expedites the development of multiphase flow regimes. The central core in Figure 2 is used to introduce air by setting the air phase velocity while water was introduced through the outer annular region by setting the water phase velocity. Both are velocity inlets.

The phase velocities are calculated as:

$$v_g = \frac{v_{sg} A}{A_g} \quad (16)$$

Table 1. Pipe sizes under investigation.

Parameter	Values	Values	Values
D_H^*	19.2	37.2	74.5
Diameter, D (m)	0.0525	0.1016	0.2032
Bend radius, R (m)	0.0762	0.1473	0.295
Length, A (m)	4	10	16
Breadth, B (m)	4	6	12
Pipe size category	Small	Intermediate	Large

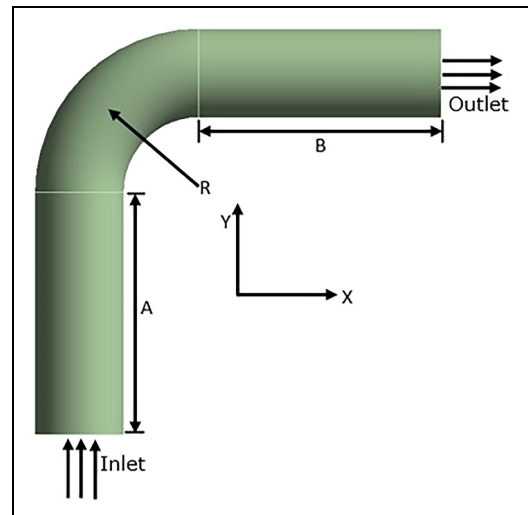


Figure 1. Boundaries of the computational domain.

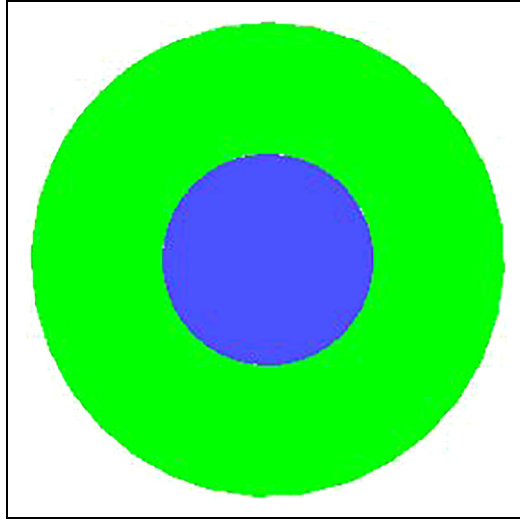


Figure 2. Domain inlet.

and

$$v_l = \frac{v_{sl}A}{A_l} \quad (17)$$

where v_g , v_l , v_{sg} , v_{sl} , A , A_g and A_l are specified gas inlet velocity, liquid inlet velocity, superficial gas velocity, superficial liquid velocity, cross-sectional area, gas area inlets and liquid area inlets, respectively.

The turbulent intensity and hydraulic diameter given as I and D_H , respectively, are the turbulence parameters which were set at the liquid and gas inlets. The turbulent intensity is calculated as:

$$I = \frac{0.16}{Re_{D_H}^{1/8}} \quad (18)$$

Re is Reynolds number and D_H hydraulic diameter. Pressure outlet is specified at the geometry's single outlet. Since the outlet is at atmospheric condition, the gauge pressure is set as zero.

CFD solution methods

The choice of the time step size is conditioned by the Courant number-based stability criterion. Based on a pre-assigned maximum allowable Courant Number of 0.25 and the smallest time spent by fluids in cells (control volume) in the region near the free surface to empty out of the cell, the present numerical algorithm automatically refines the sub-time step for the volume fraction calculation. The sub-time step size is calculated as:

$$\Delta t = C \frac{V}{\sum_{cell} U_f} \quad (19)$$

where, C is the Courant number, V is the cell volume and U_f are the outgoing fluxes in the cell. The explicit scheme is adopted for this VOF model formulation and the volume fraction cut-off and courant number are set as 1×10^{-6} and 0.25, respectively. The water and air were

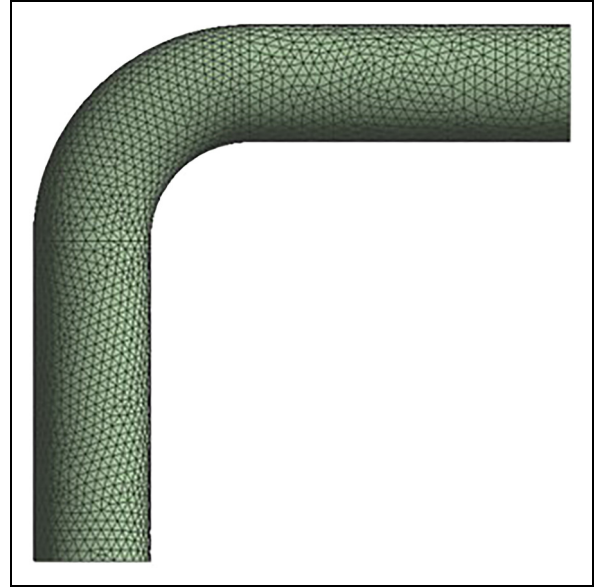


Figure 3. Structural model.

assigned as the primary phase and secondary phase, respectively. The surface tension between water and air was set at a constant of 0.0728 N/m. In addition, the SIMPLE scheme was adopted for the pressure–velocity coupling, while PRESTO and second-order upwind were implemented for pressure and momentum discretization, respectively. The volume fraction equation was solved using the Geo-Reconstruct scheme and simulation time step was fixed at 0.00001 s.

Natural frequency model

The natural frequencies and mode shapes are important parameters in the design of a structure under dynamic loading. A modal analysis determines the free vibration characteristics by solving a equation of motion for an undamped system.²⁷ The equation is given in matrix form for the elements of the structure model (Figure 3) as²⁸:

$$[\mathbf{k}]\{\mathbf{d}\} + [\mathbf{m}]\{\ddot{\mathbf{d}}\} = \{0\} \quad (20)$$

k , d m and $\ddot{d}$ are stiffness, displacement, mass and acceleration, respectively. Assuming a linear system whose motion is harmonic under free vibration, the solution of Equation (19) above becomes an Eigenvalue problem of the form:

$$(-\omega_i^2[\mathbf{M}] + [\mathbf{K}])\{\phi_i\} = \{0\} \quad (21)$$

Where ω_i is the i th natural frequency of the pipe structure in radians per unit time and ϕ_i is the displacement vector (eigenvector) representing the mode shape of the pipe structure at the i th natural frequency. $\mathbf{M}$ and $\mathbf{K}$ represent the global mass and stiffness matrices of the system. The multiphase flow in the pipe adds to the mass of the pipe depending on the global volumetric fractions of the gas and liquid flowing in the system. The natural frequency

of the pipe will change according to the added mass of each of the contained two-phase flow cases under study. The mass M is given as:

$$M = M_s + M_m \quad (22)$$

where

$$M_m = \rho_m V_p \quad (23)$$

The ANSYS Mechanical FEA tool employed in this study uses a special algorithm known as HSFLD242 3-D Hydrostatic Fluid elements to include contained fluid inside a solid. This model calculates the volume of the pipe inner channel, and the density is manually calculated and supplied as an input for each flow condition modelled. The density of fluid is given by:

$$\rho_m = \beta \rho_g + (1 - \beta) \rho_f \quad (24)$$

and

$$\beta = \frac{v_{sg}}{v_{sg} + v_{sl}} \quad (25)$$

M_s is the mass of the pipe structure and M_m represent the two-phase fluid mass, ρ_m is the mixture density, V_p is the volume of the pipe flow channel, β is the global volumetric gas fraction, ρ_g and ρ_f are the gas and liquid densities, respectively. Further details of the HSFLD242 3-D Hydrostatic Fluid Elements code can be found in ANSYS theory guide. The structural model geometry and properties are given in Table 2 and Supplemental Table 3, respectively. The pipe thicknesses are selected according to the API 5L grade line pipes. The length, breadth and bend radius are as given in Table 1 for the fluid model. Fixed end supports were defined in the model.

The test conditions and flow regime map

The flow conditions have been plotted on two vertical pipe flow regime maps based on Kaichiro and Ishii²⁹ and Taitel et al.²⁵ as shown in Figure 4(a) and (b), respectively. In Figure 4(a), Kaichiro and Ishii²⁹ flow regimes map shows the cap-bubbly flows for large pipe diameter, whereas in Figure 4(b) curves F, G, H show the transition from slug to churn depending on pipe sizes of 0.0525, 0.1016 and 0.2032 m, respectively.

Results and discussion

Mesh sensitivity and validation studies

The present CFD approach was validated and reported in an earlier study by Hossain et al.²¹ In the validation, void fraction and flow induced force signals obtained from CFD simulations were qualitatively and quantitatively compared to the experimental report of Liu et al.³ for the 0.0525 m pipe diameter. The simulation results conformed with the experimental results. Supplemental Table 4 shows the scale-up of the 0.0525 I.D. pipe to the geometrically similar intermediate and large pipes of

Table 2. Details of pipe geometry.

Properties	Small pipe	Medium pipe	Large pipe
Inner diameter, D (m)	0.0525	0.1016	0.2032
Outer diameter, D_o (m)	0.06032	0.1143	0.2191
Thickness, t (m)	0.00391	0.00635	0.00795

0.1016 m and 0.2032 m I.D. shown in Table 1. Further mesh independency studies have been carried out for each of the pipe sizes. The pipe scaling was also adopted in reporting, the largest pipe scale is 1.

Figure 5 shows a typical wall view of the meshes used in this study. Out of three different mesh sizes for each geometry, the meshes which were finally selected as a result of the mesh independence studies are 366912, 690688 and 269010 for the pipe diameters of 0.0525, 0.1016 and 0.2032 m, respectively, as shown in Figure 6(a)-(c). The geometries were created in ANSYS Design-Modeller while hexahedral meshes were created in ICEM CFD 18.0 meshing software. Supplemental Table 5 shows the refinement parameters that were adopted. The near wall cell size was predicted using the expression:

$$\Delta y = \frac{y^+ \mu}{U_\tau \rho} \quad (26)$$

and

$$U_\tau = \sqrt{\frac{\tau_w}{\rho}} \quad (27)$$

$$\tau_w = 0.5 C_f \rho U_\infty^2 \quad (28)$$

U_τ is the friction or shear velocity and τ_w is the wall shear stress. For internal pipe flows, friction coefficient, C_f is given based on Blasius equation as:

$$C_f = 0.079 Re_d^{-0.25} \quad (29)$$

where Δy and U_∞ are the first cell height and the free stream velocity, respectively. y^+ values between 30 and 100 were chosen.

For mesh independence study, the slug flow condition was modelled as a homogenous mixture. Velocity profiles shown in Figure 7(a) and (b) were obtained in the 0.1016 and 0.2032 m I.D. pipes. The figures show that the velocity profile at planes positioned at 11.6 and 15.4 m, respectively, from the entry upstream of the bend for all three meshes in each case are identical and they represent typical velocity profile of fully developed turbulent flows in circular pipes.

Typical flow patterns obtained by the simulation

Figure 8 and Supplemental Figure 9 present the contour plots and PDFs of typical slug and churn flow patterns

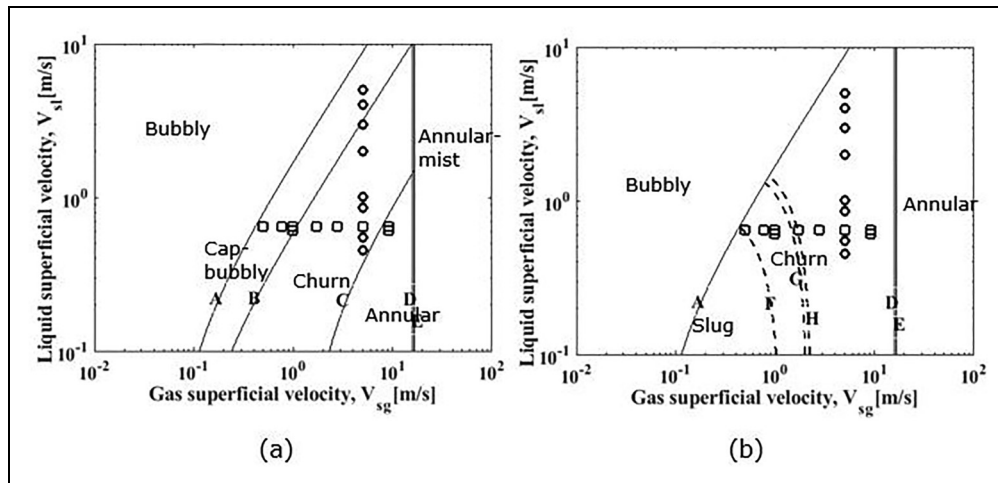


Figure 4. (a) Kaichiro and Ishii²⁹ flow regime map including Schlegel et al.³⁰ criterion for cap-bubbly to churn turbulent flow transition in large pipes, (b) Taitel et al.²⁵ entry length phenomenon for slug to churn flow transition.

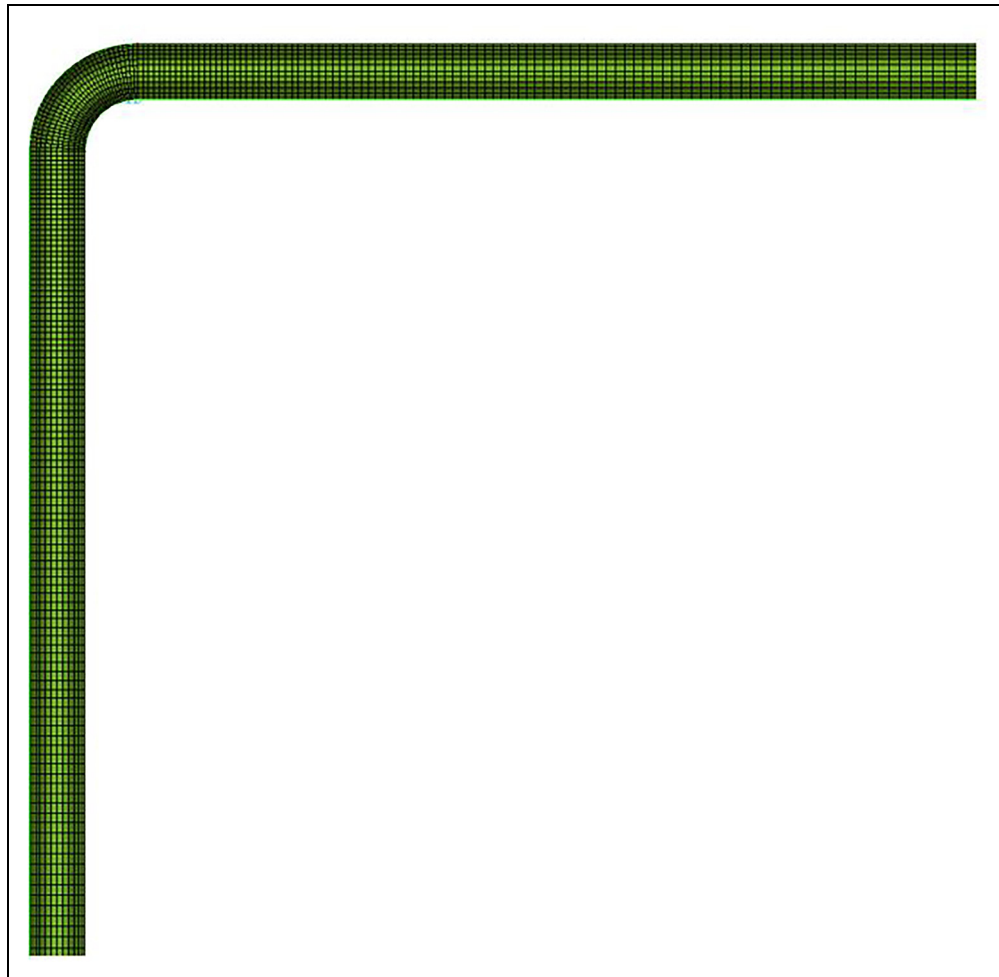


Figure 5. Meshing on pipe wall.

observed in the three pipe sizes. Both flow conditions were validated for the pipe of 0.0525 m I.D. in the CFD investigation carried out by Hossain et al.²¹ Findings were compared to the experimental results of Liu et al.³ Figure 8 and Supplemental Figure 9(b) represent the

PDFs of the void fraction fluctuation extracted at a cross-section above 70D from the pipe inlet. Figure 8(b) shows that the two peaks in the PDFs of void fraction in the small pipe became almost equal while in the larger pipes the leading peak representing low void fraction of liquid

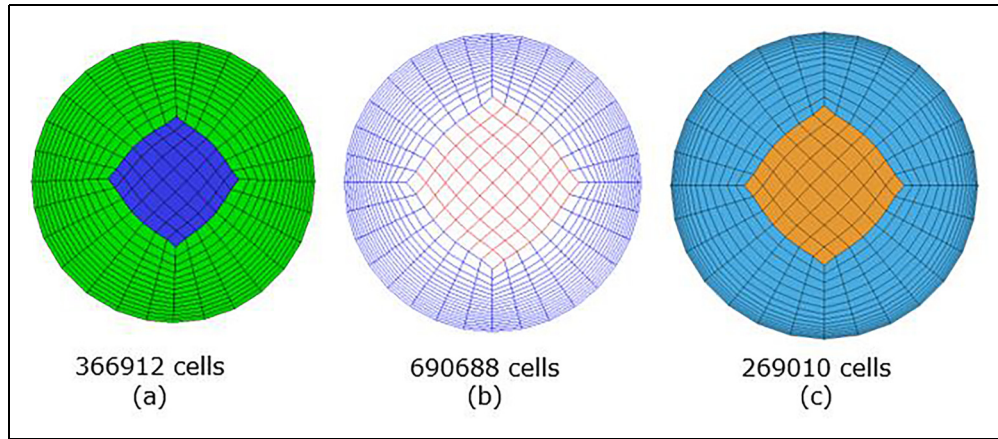


Figure 6. Pipe meshes for I.D. (a) 0.0525 m, (b) 0.1016 m and (c) 0.2032 m.

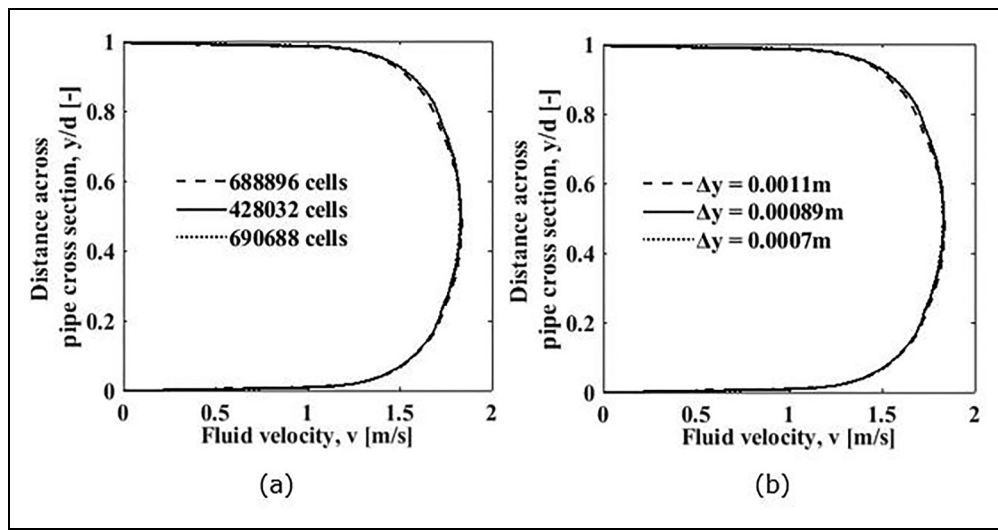


Figure 7. (a) and (b) show the predicted velocity profiles for different mesh sizes in pipes of 0.1016 and 0.2032 m, respectively.

slug containing small bubbles was higher than the trailing peak which represents cap bubbles. These behaviours are consistent with reported experimental observations of slug and cap bubbly flow regimes characteristics.^{30–32} The contour plot in Figure 8 shows that the liquid slug could still bridge the pipe cross-section. On the other hand, the contour plot in Supplemental Figure 9 shows that the liquid slugs could rarely bridge the pipe cross-section. The large liquid wave structures which are characteristic of churn flows are also visible in Supplemental Figure 9(a) and the observation is consistent with the findings in literature.^{31–34} The PDFs in Supplemental Figure 9(b) all show peaks above 0.8 in the three pipe sizes. In summary, the CFD method was able to reproduce the basic physical behaviours of the slug, cap-bubbly and churn turbulent flow regimes expected in the pipes. Slug flow regimes which were observed in the vertical pipe section in small pipes transitioned into cap-bubbly flow in large pipes.

Two-phase flow induced force

The fluctuating forces acting on the elbow of the pipe were calculated using equations (29) to (31) based on a

control volume around the elbow. The x is in the horizontal direction, while the y is in the vertical direction (see, Figure 1).

$$F_x(t) = \dot{m}(t)V(t) + p(t)A$$

– at the exit plane of the bend (30)

$$F_y(t) = -\dot{m}(t)V(t) - p(t)A$$

– at the inlet plane of the bend (31)

$$\dot{m}(t) = [\alpha(t)\rho_g + (1 - \alpha(t))\rho_l]AV(t) \quad (32)$$

F_x , F_y , $\dot{m}$, V , t , p , A , ρ_g and ρ_l are force in x -direction, force in y -direction, mass flow rate, velocity, time, pressure, pipe cross-section area, gas density and liquid density. Supplemental Figures 10–18 show the simulated time-histories of force fluctuations and the corresponding PSDs of the fluctuations in each case for the flow regimes identified in Figure 4. Supplemental Figures 10(a) to 14(a) show that force fluctuations in the 0.1016 m I.D. pipe were generally within -100 to 100 N while -500 to 500 N was observed in the 0.2032 m I.D. pipe

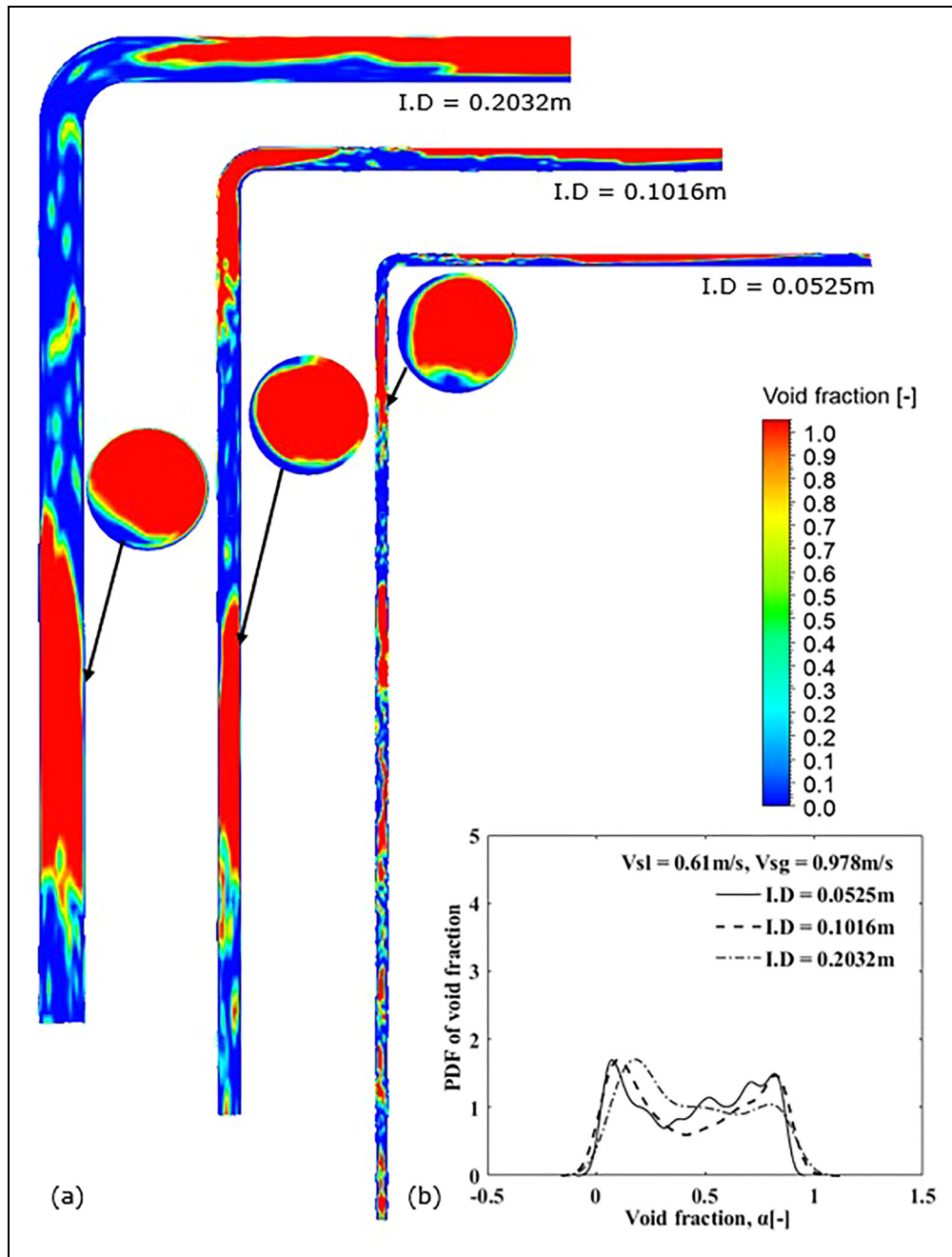


Figure 8. (a) Contour of air volume fraction and (b) PDF for $V_{sl} = 0.61$ m/s and $V_{sg} = 0.978$ m/s in the three pipe sizes.

for moderate slug flow up to $V_{sg} = 1.7$ m/s. At $V_{sg} > 1.7$ m/s representing unstable slug and churn flows, the fluctuation ranges increased to about -250 to 250 N in the 0.1016 m I.D. pipe and -2000 to 2000 N in the 0.2032 m I.D. pipe due to the increase of mass flow rate and hence, momentum flux with the increase of pipe diameter. The only exception is at $V_{sg} = 5$ m/s which exhibited reduced force fluctuation which can be attributed to slug-churn flow regime transition. This result conforms to a reported experimental measurement of -500 to 1000 N for $V_{sg} = 10.45$ m/s and $V_{sl} = 1$ m/s in a 90° pipe bend of 0.154 m I.D.⁹

The y -components of force fluctuations were consistently higher than the observed x -component fluctuations although, the magnitude of the difference reduced as gas

superficial velocity increased towards churn flow regime ($V_{sg} = 9.04$ m/s). This observation is consistent with findings reported in Hossain et al.²¹ for the pipe of 0.0525 m I.D. The high amplitude of F_y fluctuation is attributed to the impact of liquid slugs on the bend in the y -direction while the lower amplitude of F_x which is more prominent in the moderate slug flow cases up to V_{sg} of 1.7 m/s (Supplemental Figures 10 to 14) is as a result of the change in flow pattern into stratified/stratified wavy flow downstream of the bend.

In both pipes, Supplemental Figures 10(b) to 18(b) show that higher frequencies greater than 2 Hz became more prominent as V_{sg} increased beyond 1.7 m/s except at V_{sg} of 5 m/s which showed less force fluctuation

energy beyond 2 Hz. This behaviour was also present in the 0.0525 m I.D. pipe investigated in Hossain et al.²¹ However, in contrast to the findings in the case of 0.0525 m I.D. pipe, force fluctuations PSD spread over larger frequency ranges especially for the churn flows of $V_{sg} = 9.04$ m/s. The PSD plots show that the strength of force fluctuation is significantly higher in the 0.2032 m I.D. pipe than in the 0.1061 m I.D. pipe.

Supplemental Figures 19 and 20 show the predominant frequency and the RMS of force fluctuations in the x and y direction, respectively, for all three pipe sizes. The predominant frequency is identified as the peak frequency corresponding to the maximum force fluctuation strength in flow directions. Supplemental Figures 19(a) and 20(a) show that the predominant frequency increase gradually followed by a drop as gas superficial velocity increased from slug flow regime to churn flow regime in the 0.0525 and 0.2032 m I.D. pipes. A more irregular behaviour was rather observed for the 0.1016 m I.D. pipe especially within the slug flow regime. In addition, the present single 90° bend study showed that at a constant liquid superficial velocity of 0.642 m/s, the predominant frequencies of force fluctuation were at a maximum of 1.7 Hz in the 0.0525 m (small) pipe and reduced to 1 and 0.8 Hz in the 0.1016 and 0.2032 m I.D. pipes, respectively. However, the other important peak frequencies which exist beyond 2 Hz especially in churn flow regimes should be taken into consideration in the design and monitoring of pipe network of similar dimensions to avoid the occurrence of resonance.

The RMS of force fluctuations in Supplemental Figures 19(b) and 20(b) showed similar behaviour with increasing superficial gas velocities in the three pipe sizes. At constant liquid superficial velocity, the RMS increased monotonically with gas superficial velocity within slug flow regime. However, at $V_{sg} = 5$ m/s which coincides with the transition region to churn flow regime, the RMS of force fluctuation dropped suddenly in the larger pipes, before increasing significantly again in the churn flow regime. Also, the RMS of force fluctuation were significantly (>5 times) higher in the 0.2032 m I.D. pipe compared to the 0.1016 and 0.0525 m I.D. pipes despite a size scale ratio of only 2 and 4, respectively. The maximum values of RMS were observed in churn flow regime and values of 450, 72 and 18.5 N were calculated for the large, intermediate, and small pipes, respectively.

Momentum flux and pressure fluctuation

Supplemental Figures 21 and 22 present the contributions of the momentum flux and fluid pressure fluctuations to the calculated force fluctuations. The momentum and pressure fluctuations arise from the alternating nature of liquid and gas phases in slug and churn flow through pipe bend control volume. The bend control volume analysis that was adopted in the present study to calculate force at the bend was based on momentum balance theory also implemented by Liu et al.³ A control volume created at the bend starts at a plane positioned at the immediate upstream vicinity of the bend forming the

inlet of the control volume. The control volume terminates at the immediate downstream region of the bend forming the control volume exit. Pressure and momentum are calculated on the inlet and outlet surfaces at each time step and the values depend on the fluid phase flowing through the surface at the time. Similar to the findings of Hossain et al.²¹ for a small pipe of 0.0525 m I.D., Supplemental Figure 19(a) shows that the RMS of momentum flux fluctuation is significantly correlated to the RMS of force fluctuation in the x -direction for the intermediate and large pipes examined in the present study. This was not the case in the y -direction. Supplemental Figure 19(b) shows a less significant correlation between the momentum flux fluctuation and the force fluctuation in the intermediate and large pipes compared to the small pipe. This is more pronounced at lower force and momentum values corresponding to slug and cap-bubbly flow regimes where $V_{sg} \leq 0.978$ m/s and $V_{sl} \leq 0.642$ m/s. Supplemental Figure 20(b) shows that the deficiency in the RMS of momentum flux fluctuation corresponding to the cap-bubbly flow regime in large pipes was compensated by the pressure force fluctuation which correlated more to the force fluctuation in y -direction in the larger pipes compared to the small pipe.

Performance of Riverin et al.¹¹ correlation for non-dimensional RMS of excitation force in both small and large pipes

Supplemental Figures 23 (a) and (b) present the comparisons of the non-dimensional RMS of force fluctuation for the three pipe sizes to the model by Riverin et al.¹¹ In Supplemental Figure 21 (a), the constant C appearing in equation (32) is left as its original value of 10. In Supplemental Figure 21 (b), the correlation has been modified by setting the value of the constant C to be 20. The modified correlation is given as:

$$\overline{F_{rms}} = \frac{F_{rms}}{\rho V_l^2 \left(\frac{\pi D^2}{4} \right)} = 20 We^{-0.4} \quad (33)$$

$\overline{F_{rms}}$, F_{rms} and We represent the non-dimensional RMS of force fluctuation, RMS of force fluctuation and Weber number. Most of the CFD predicted data in Hossain et al.²¹ for the small pipe of 0.0525 m I.D. fall within the $\pm 50\%$ accuracy margin of the correlation by Riverin et al.¹¹ However, Supplemental Figure 21(a)(i) shows that close to 70% of the predicted data in the 0.1016 m I.D. fall outside the $\pm 50\%$ range while about 90% of predicted data in the 0.2030 m I.D. pipe fall outside the range. In addition, Supplemental Figure 21(a)(ii) in which the predicted data have been grouped according to the global volume fraction β shows that pipe diameter has a stronger effect on $\overline{F_{rms}}$ compared to β . Generally, the correlation could not estimate about 50% of the combined 35 flow case studies across the three pipe sizes. The experimental findings by Belfroid et al.⁶ for a large pipe of 0.1524 m I.D. matches the present study findings. Most of their experimental data also fell outside the

$\pm 50\%$ even for $40 < \beta < 80$ as was equally observed in the present study. On the other hand, Supplemental Figures 23(b)(i) and (ii) show that the modified correlation having constant $C = 20$ is able to estimate about 70% of the combined CFD/analytical predicted $\overline{F_{rms}}$ across the three pipe sizes within $\pm 50\%$ accuracy range.

Effect of flow pattern on natural frequency

Supplemental Figures 24(a) to (c) present the natural frequencies of the three similar pipe sizes. The added mass effect due to contained fluid is captured by defining the global volumetric gas fraction at the pipe inlet. Generally, as the gas fraction increases corresponding to increasing gas superficial velocities (V_{sg}) represented by the increasing trend of V_{sg}/V_{sl} in Supplemental Figure 24, the natural frequencies of the first three modes of vibration increases, approaching the highest value corresponding to gas volumetric fractions (β) of 0. Based on their experiment, Wang et al.²⁰ also reported that increasing gas volume fraction at inlet had similar effect on the first three natural frequencies of an acrylic material pipeline riser of 0.0514 m I.D. and thickness of 0.0058 m conveying gas-liquid two-phase flow. However, the present study shows that the increase in natural frequencies with gas superficial velocities and volume fraction were most significant in the smallest pipe of $\frac{1}{4}$ scale than the $\frac{1}{2}$ and 1.0 scale sized pipes. Frequencies as low as 1 Hz were recorded in the 1.0 scale pipe while the lowest frequency in the $\frac{1}{2}$ scale pipe is ~ 2 Hz.

Resonance risk assessment

Risk of resonance in the three pipe sizes, subjected to identical internal multiphase flow conditions were examined. The 1.0 scale pipe presented the highest resonance occurrence risk. Supplemental Figure 25(a) to (c) presents the comparison of the first three natural frequencies of the pipe bends with fixed supports (representing modes 1–3) as a function of gas superficial velocity and the dominant frequencies of force fluctuation. In Supplemental Figure 23(c), the dominant frequencies of force fluctuation were close to the first natural frequency of the pipe structure with and without contained fluid at gas superficial velocities of 1.7 and 2.765 m/s. Generally, the risks of resonance were most likely within slug/cap bubbly and churn turbulent flow regimes up to V_{sg} of 5 m/s.

In summary, although the risk of resonance in the $\frac{1}{4}$ scale (Supplemental Figure 23(a)) is insignificant, further studies might show frequencies of pipe deformation or displacement coinciding with natural frequencies or high frequency stress circles which could also reduce the fatigue life of the structure.

Conclusion

A numerical investigation to evaluate the effect of diameter on MFIV characteristics and the effect of internal two-phase flow on the resonance behaviour of three different sizes of geometrically similar pipe bends have

been carried out. The significant findings in this study are summarised as follows:

1. At constant liquid superficial velocity, the predominant frequency of force fluctuation in flow direction increases gradually with the increase of gas superficial velocity within slug flow regime followed by a drop as the gas superficial velocity increases to churn flow regime in the 0.0525 m ($\frac{1}{4}$ scale) and 0.2032 m (1 scale) I.D. pipes. A more irregular behaviour was present in the 0.1016 m I.D. (intermediate) pipe.
2. A very good corroboration of RMS of resultant force fluctuations with Riverin et al.¹¹ correlation was observed for the $\frac{1}{4}$ scale pipe where the constant C is 10. However, when the larger pipe sizes were considered, the correlation using $C = 10$ could not predict the non-dimensional RMS of forces well. A better prediction was obtained using $C = 20$.
3. In all cases, the presence of internal two-phase fluid changed the natural frequencies of pipe. This change was more significant for small pipes. Increasing the gas volume fraction increases the natural frequencies of the pipe, especially in the small pipe.
4. However, the risk of resonance was higher in the large pipe since the observed low natural frequencies were closely matched with the low peak frequencies of force fluctuations.

Declaration of conflicting interests

The author(s) declared no potential conflicts of interest with respect to the research, authorship, and/or publication of this article.

Funding

The author(s) disclosed receipt of the following financial support for the research, authorship, and/or publication of this article: This work was supported by the Commonwealth Scholarship Commission in the UK for Nkemjika Mirian Asiegbu's PhD study.

ORCID iDs

Nkemjika Mirian Asiegbu  <https://orcid.org/0000-0002-4461-5651>

Mamdud Hossain  <https://orcid.org/0000-0002-9492-2721>

Supplemental material

Supplemental material for this article is available online.

References

1. Miwa S, Mori M and Hibiki T. Two-phase flow induced vibration in piping systems. *Prog Nucl Energy* 2015; 78: 270–284.
2. Parameshwaran R, Dhulipalla SJ and Yendluri DR. Fluid-structure interactions and flow induced vibrations: a review. *Procedia Eng* 2016; 144: 1286–1293.
3. Liu Y, Miwa S, Hibiki T, et al. Experimental study of internal two-phase flow induced fluctuating force on a 90 elbow. *Chem Eng Sci* 2012; 76: 173–187.

4. Isao K and Mamoru I. Drift flux model for large diameter pipe and new correlation for pool void fraction. *Int J Heat Mass Transf* 1987; 30: 1927–1939.
5. Nennie E and Belfroid S. Multiphase forces on bend structures. In: *ICMF-2016, 9th International Conference on Multiphase Flow, 22–27 May 2016, Firenze, Italy* 2016, TNO.
6. Belfroid SP, Nennie E, Wijhe A, et al. Multiphase forces on bend structures—overview of large scale 6” experiments. In: *11th International Conference on Flow-Induced Vibration*, Hague, Netherlands, 2016.
7. Emmerson P. CFD validation of flow induced forces in 6” piping caused by slugging flow. In: *11th International Conference on Flow-Induced Vibration*, Hague 2016.
8. Emmerson P, Lewis M and Barton N. Influence of multiple bends for multiphase flow induced forces in 6” piping: measurements and CFD analysis. In: *11th International Conference on Flow-Induced Vibration*, Dublin 2016.
9. Pontaza JP, Kreeft JJ, Abuali B, et al. Flow-induced forces in piping due to multiphase flow prediction by CFD. *11th International conference on flow-induced vibration, FIV* 2016. Hague 2016.
10. Belfroid S, Nennie E and Lewis M. Multiphase forces on bend structures—influence of upstream disturbance. In: *10th North American Conference on Multiphase Technology* 2016, OnePetro.
11. Riverin J, De Langre E and Pettigrew M. Fluctuating forces caused by internal two-phase flow on bends and tees. *J Sound Vib* 2006; 298: 1088–1098.
12. Yamano H, Tanaka M-a, Kimura N, et al. Development of flow-induced vibration evaluation methodology for large-diameter piping with elbow in Japan sodium-cooled fast reactor. *Nucl Eng Des* 2011; 241: 4464–4475.
13. Yamano H, Tanaka M, Murakami T, et al. Unsteady elbow pipe flow to develop a flow-induced vibration evaluation methodology for Japan Sodium-Cooled Fast Reactor. *J Nucl Sci Technol* 2011; 48: 677–687.
14. Emmerson P, Lewis M and Barton N. Improving boundary conditions for multiphase CFD predictions of slug flow induced forces. In: *17th International Conference on Multiphase Production Technology* 2015, OnePetro.
15. Pontaza JP, Menon RG, Okeremi A, et al. Flow-induced vibrations of high gas rate well jumpers: tees vs. bends. In: *International Conference on Offshore Mechanics and Arctic Engineering* 2013, p.V007T008A082. American Society of Mechanical Engineers.
16. Pontaza JP, Abuali B, Brown GW, et al. Flow-induced vibrations of subsea piping: a screening approach based on numerical simulation. In: *SPE Offshore Europe Oil and Gas Conference and Exhibition* 2013, OnePetro.
17. Takaya S, Fujisaki T and Tanaka M. Numerical Analysis of Flow-Induced Vibration of Large Diameter Pipe With Short Elbow. In: *Pressure Vessels and Piping Conference* 2017, p.V004T004A006. American Society of Mechanical Engineers.
18. Sanchis A and Jakobsen KG. Flow-induced vibrations in subsea pipework. In: *Proceedings of the 10th International Conference on Flow-Induced Vibration (& Flow-Induced Noise), FIV* 2012 Dublin, Ireland, 3–6 July 2012.
19. Kim J and Srinil N. 3-D numerical simulations of subsea jumper transporting intermittent slug flows. In: *International Conference on Offshore Mechanics and Arctic Engineering* 2018, p.V002T008A028. American Society of Mechanical Engineers.
20. Wang L, Yang Y, Li Y, et al. Resonance analyses of a pipeline-riser system conveying gas-liquid two-phase flow with flow-pattern evolution. *Int J Press Vessel Pip*. 2018; 161: 22–32.
21. Hossain M, Chinenye-Kanu NM, Droubi GM, et al. Investigation of slug-churn flow induced transient excitation forces at pipe bend. *J Fluids Struct* 2019; 91: 102733.
22. Nichols B and Hirt C. Methods for calculating multidimensional, transient free surface flows past bodies. In: *Proceedings of the First International Conference on Numerical Ship Hydrodynamics* 1975.
23. Hirt CW and Nichols BD. Volume of fluid (VOF) method for the dynamics of free boundaries. *J Comput Phys* 1981; 39: 201–225.
24. Launder BE and Spalding DB. The numerical computation of turbulent flows. *Comput Methods Appl Mech Eng* 1974; 2: 269–289.
25. Taitel Y, Bornea D and Dukler A. Modelling flow pattern transitions for steady upward gas-liquid flow in vertical tubes. *AIChE J* 1980; 26: 345–354.
26. Parsi M, Agrawal M, Srinivasan V, et al. Assessment of a hybrid CFD model for simulation of complex vertical upward gas-liquid churn flow. *Chem Eng Res Des* 2016; 105: 71–84.
27. Meirovitch L. *Fundamentals of vibrations*. Long Grove, IL: Waveland Press, 2001.
28. Pavlou DG. *Essentials of the finite element method: for mechanical and structural engineers*. London: Academic Press, 2015.
29. Kaichiro M and Ishii M. Flow regime transition criteria for upward two-phase flow in vertical tubes. *Int J Heat Mass Transf* 1984; 27: 723–737.
30. Schlegel JP, Sawant P, Paranjape S, et al. Void fraction and flow regime in adiabatic upward two-phase flow in large diameter vertical pipes. *Nucl Eng Des* 2009; 239: 2864–2874.
31. Costigan G and Whalley PB. Slug flow regime identification from dynamic void fraction measurements in vertical air-water flows. *Int J Multiph Flow* 1997; 23: 263–282.
32. Lowe DC and Rezkallah KS. Flow regime identification in microgravity two-phase flows using void fraction signals. *Int J Multiph Flow* 1999; 25: 433–457.
33. Montoya G, Lucas D, Baglietto E, et al. A review on mechanisms and models for the churn-turbulent flow regime. *Chem Eng Sci* 2016; 141: 86–103.
34. Da Riva E and Del Col D. Numerical simulation of churn flow in a vertical pipe. *Chem Eng Sci* 2009; 64: 3753–3765.

Appendix

Notation

A	pipe cross-sectional area, m ²
A_g	surface area of gas inlet, m ²
A_l	surface area of liquid inlet, m ²
C_μ , $C_{1\epsilon}$, $C_{2\epsilon}$	coefficients in approximated turbulent transport equations
C	constant (Riverin correlation)
C_f	friction coefficient
d	nodal displacements, m
$\ddot{d}$	acceleration, m/s ²
D	pipe diameter, m

D_H	non-dimensional hydraulic diameter
D_H^*	non-dimensional hydraulic diameter
E	empirical constant given as 9.793
F	external body forces, N
F_x	force in x -direction, N
F_y	force in y -direction, N
$\overline{F_{rms}}$	non-dimensional RMS of force fluctuation
F_{rms}	RMS of force fluctuation
g	gravity, m/s^2
k	turbulent kinetic energy
m	mass, kg
$\dot{m}$	mass flow rate, m/s
$\bar{n}_w$	unit vectors normal to the wall
p	pressure, Pa
Re	Reynolds number
$\bar{t}_w$	unit vectors tangential to the wall
t	time, s
$\bar{u}$	velocity component in $i, j = 1, 2$ or 3 directions, m/s
U_p	time averaged velocity of fluid at point P , m/s
U_∞	free stream velocity, m/s
U_τ	friction or shear velocity, m/s
$\vec{v}_m$	mixture velocity, m/s
v_f	liquid phase velocity, m/s
Vsg (v_{sg})	gas superficial velocity, m/s

Vsl (v_{sl})	liquid superficial velocity, m/s
$V(t)$	instantaneous velocity, m/s
V_t	mixture velocity, m/s
We	Weber number
Δy	first mesh cell height, m

Greek letters

α	volume fraction of one phase
$\tilde{\alpha}_g^A$	area averaged void fraction
ϵ	energy dissipation rate
θ_w	contact angle
κ_l	surface curvature of the liquid droplet
κ_g	surface curvature of the gas bubble
κ	Von Karman constant given as 0.4187
μ	dynamic viscosity, kg/m s
μ_f	dynamic viscosity of liquid, kg/ms
ρ	density, kg/m^3
ρ_f	liquid density, kg/m^3
ρ_m	mixture density, kg/m^3
ρ_g	gas density, kg/m^3
ρ_l	water density, kg/m^3
$\Delta\rho$	density difference between gas and liquid, kg/m^3
σ	surface tension, N/m
$\emptyset$	arbitrary flow property
τ_w	wall shear stress, N/m^2