Proceedings of the ASME 2017 International Mechanical Engineering Congress and Exposition
IMECE2017
November 3-9, 2017, Tampa, Florida, USA

IMECE2017-71038

NUMERICAL ANALYSIS OF THE PREDICTION OF THE TWO-PHASE FLOW RATE BY MEASURING VIBRATION OF PIPELINES

Ahmed M. Nagib Elmekawy
Department of Mechanical Engineering
Alexandria University
Alexandria, Egypt 11432
Email: a.nagib@alexu.edu.eg

Mohamed A. Shabara
Department of Mechanical Engineering
Alexandria University
Alexandria, Egypt 11432
Email: shabara91@gmail.com

Hassan Elgamal
Department of Mechanical Engineering
Alexandria University
Alexandria, Egypt 11432
Email: ha_elgamal@yahoo.com

Bassuny El-Souhily
Department of Mechanical Engineering
Alexandria University
Alexandria, Egypt 11432
Email: bmelsouhily@alexu.edu.eg

ABSTRACT

Multi-phase flow is encountered in multiple industrial disciplines including oil and gas industry starting from the reservoirs, production tubes, well heads, separators and transportation systems comprehending risers & onshore/offshore transportation pipelines. The complex nature & instability of slug flow where pressure, temperature, velocities and flow rates oscillate lead to high fatigue loads on mechanical elements in pipelines. The prediction of the two-phase flow pattern inside pipelines is crucial in extending the life of the pipeline. This paper proposes the first numerical approach for predicting the vibration of a three-dimensional pipeline with an elbow due to two-phase flow using one-way fluid-structure interaction technique using the commercial software ANSYS. The unsteady Reynolds-averaged Navier-Stokes equations with three turbulent models are used to model the fluid domain. The finite element analysis is used to model the pipeline by using shell elements. The two-phase mixture superfacial velocities inside the pipeline were identified by measuring the acceleration response of the surface of the pipe. Modal analysis is performed to check the variation of the pipeline natural frequency with the water content in the flow. The simulation results were validated with experimental data. Based on the simulation results, a numerical method for the measuring of the two-phase flow rates is proposed.

NOMENCLATURE

C	Damping matrix
Ca	Capillary number
CSF	Continuum Surface Force
F	Generalized hydrodynamic load tensor imported from the CFD solver
FSI	Fluid Structure Interaction
G	Gas
H	Holdup
K	Stiffness matrix
L	Liquid
M	Mass matrix
P	Static pressure
Re	Reynolds number
S	Mass source
T	Viscous stress

U	Volume flux based on normal velocity
V	Volume of the cell
$\dot{m}$	Mass transfer
f	Face value of a certain volume fraction
f_c	Capillary force
f_g	Gravitational force
g	Gravitational acceleration
n	index for previous time step
$n+1$	Index for current time step
t	Time
u	Velocity vector
p	p^{th} phase
q	q^{th} phase
α	Phase volume fraction
μ	Dynamic viscosity
ρ	Density
σ	Surface tension
ζ	Generalized structural tensor displacement

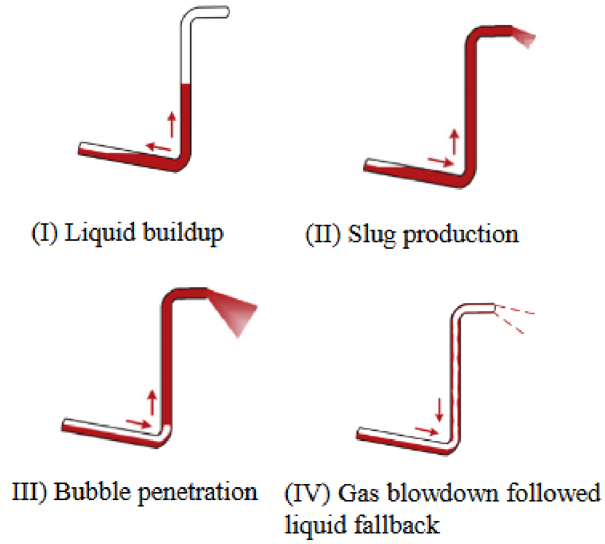

FIGURE 1. TERRAIN INDUCED SLUGGING IN RISERS

INTRODUCTION

Numerous problems were reported in industry due to the generation of unexpected flow slugging in pipelines and piping systems. Published data suggests that up to 20% of the pipework failures that resulted in hydrocarbon release may have caused by vibration induced fatigue. Santana et al. [1] reported an accident that occurred in Prudhoe Bay Oil Field, Kuparuk River, North Slope, Alaska where ARCO (the operating partner) suffered from flow slugging problem in its piping systems after start-up in 1981 resulting in cracked nozzle in a vessel, this drove ARCO to expensive modifications.

Also terrain-induced slugging problem reported by WEPCO in Abu Qir Gas Fields in a gas turbine/compressor set in the on-shore compression station. The compressor had a wet mechanical seal system that suffered from a problem with the inclination of the riser of the degassing tank vent. This change in the riser's inclination caused oil to accumulate upstream the riser leading to a complete blockage in the flowing gases path. This blockage resulted in excessive pressure build up until the gases inside the lube oil and the degassing tanks reached the self-ignition point causing explosions. These explosions forced oil slugs to flush outside the riser. Part of the oil falls back to the riser base to create new blockage and the same cycle was repeated every start causing the engine to shutdown and hence production loss as discussed in Fig.1. This sequence can also occur in oil & gas production risers [2], and transportation pipelines [3] leading to cracks and failures in the upstream piping system.

According to Hill and Wood [4] slugs of about 600 meters were generated in a pipeline causing difficulty in the slug catchers operation, with high separator level shutdowns and hence production loss and increasing the plant downtime. In 1999, Mores & Fairhurst [5] reported that BP has been trying to predict the vortex induced vibration related fatigue in order to determine the required vortex induced vibration (VIV) suppression devices, and that the slugging phenomenon can cause significant impacts on deep water risers that are more than 800 meters depth. Even though many techniques were used to inhibit the flow slugging [6] or suppress its effects [7]. There are cases where flow slugging is unavoidable during plants operation such as start-ups, shutdowns and during pipeline pigging jobs. In deep water operating reservoirs where pressure & temperature fluctuations can become significant causing several production changes [8] that may require operating in off-design conditions. Also wax and hydrates formation in pipeline in the presence of flow slugging can cause high vibration and failure problems to pipelines.

Many experimental investigations were carried out [9–15] to quantify the induced forces on piping elements due to multiphase flow using various test benches for different flow chemical & physical properties. Few numerical studies were performed to quantify these forces such as the experiment carried out in 2010 by Belfriod et al. [16] to measure the forces and acceleration responses of various horizontal piping arrangements, and found that the forces increased four orders in magnitude for slug flow compared to single phase flow with same velocity, and also found that the annular flow regime has the lowest impact on these mechanical components. Tay & Thrope [17] carried out their experiment to study the effect of the fluid's physical properties particularly viscosity and surface tension on the maximum forces acting on a pipe bend and on liquid holdup and compared their results to the one-dimensional Piston Flow Model ”PFM” that was proposed in a previous work presented by Tay & Thrope [12]. This PFM is a control volume analysis that simulates the slugs as alternate pistons having speed that is similar to the slug speed in which the unsteady momentum equation can be applied. This model succeeded in providing accurate numerical-analytic results for calculating the maximum hydrodynamic force acting on an elbow. An & Su [18] used the Generalized Integral Transform Technique (GITT) to investigate the dynamic behavior of pipes conveying two-phase flow numerically. This approach was confirmed to give similar results as Monette & Pettigrew [19].

In 2009, Gama et al. [20] suggested a low-cost technique inspired by Evans et. al [21] work for determining the two-phase flow rate and the liquid content in the mixture that does not require separation of phases. Experimental investigations were carried out on L & U shaped pipe segments and the acceleration responses were measured in different planes. Quadratic correlations relating the RMS acceleration response and two-phase fluid flow mixture velocity were deduced. Also experimental and numerical studies were performed to relate the pipe natural frequency to the liquid content.

Various numerical studies have been performed to simulate the slug flow regime in vertical, horizontal and inclined piping systems such as the investigation carried out by Frank [22] using the CFX code to study the effect of using periodic inlet & outlet boundary conditions as well as the effect of the pipe length in horizontal slug flows in circular pipes. It was found that the wall friction of the liquid phase is the main source of the flow regime transition from segregated to intermittent (slug) flow regime. It was also observed that the speed and place of the slugs generation strongly depends on the agitation and perturbation of the inlet boundary conditions. In numerical simulations with prescribed constant inlet velocities and volume fractions without any transient perturbation of the free surface between water and air, very long pipe segments were necessary in the numerical simulations in order to observe the formation of liquid slugs. In 2012, Czapp et al. [23] used High-speed SPIV (Stereo Particle Image Velocimetry) with LIF (Laser Induced Fluorescense) applied around the pipe geometry in order to capture the slug initiation and propagation. Furthermore, CFD simulations were run using OpenFOAM, in order to shorten the time of simulation, the slug generation was performed by superimposing a sinusoidal wave over the superficial water inlet velocity for the first half second. Afterwards the boundary conditions (superficial water & air velocities) were kept constant during the measurement. Lu [24] carried out experimental and numerical simulations for horizontal gas-liquid two-phase slug flow in circular and rectangular pipes, the numerical study was performed using six CFD commercial codes (TRIOMPH, FLUENT, STAR-CD, LedaFlow, TransAT and CFX). Constant boundary conditions were used for all the simulations. This caused CFX solver to fail in prediction of the flow instability by either not transforming the stratified to slug flow or the flow instability occurred at a further distance downstream compared to the experimental data. For the rectangular cross sectional area, it was noticed that when the slip between phases exceeds a certain limit, FLUENT failed to predict the flow instability and the flow remained stratified irrespective to the turbulence model used.

In 2016, Fangqiu et al. [25] studied the flow induced vibration of sub-sea span pipeline but without considering the influence of external flow, by performing two way coupling between the CFX and ANSYS Mechanical, the vibration displacement, frequency and von Mises stress response were monitored at the pipeline center, it was observed that the flow induced vibration is directly proportional to air volume fraction. However, they did not validate their model. Also, the utilized time step was too large (0.1 sec) which could affect the accuracy of the calculated stress and vibration responses adversely when modeling fluid slugging phenomenon.

In this work, we propose a new three dimensional numerical model using the one-way Fluid Structural Interaction technique that can predict slug flow formation and growth using the CFD solver FLUENT, then apply the resulted forces to the Transient ANSYS Mechanical to quantify the induced vibration on an elbow conveying water & air mixture. Also, numerical modal analysis is performed to verify the sensitivity of the pipe natural frequency to void fraction variation. Both numerical models were validated with experimental data reported by Gama et al. [20] and provided encouraging results.

MODEL FORMULATION

Physical Model Formulation

Considering a horizontal acrylic standard long elbow ($R/D_i = 3$) connected to two acrylic pipes with inner & outer diameter 25.4 & 31.8 *mm* respectively. The pipes upstream & downstream the elbow are clamped at distances 720 & 810 *mm* measured from the elbow center; respectively. An accelerometer is mounted on the pipe elbow center perpendicular to the pipe plane to measure flow induced vibration. This piping system is subjected to water/air two-phase internal flow. This setup is identical to the L-pipe test section used by Gama et al. [20] as shown in Fig. 2.

In laboratory experiments the phases are mixed at a cross section further upstream of the test section to guarantee the generation of fully developed flow entering the test section. However, due to the high flow velocities and the computational resources it is not applicable to model the whole pipe length

FIGURE 2. PIPING SYSTEM USED IN THE NUMERICAL STUDY

required for developing fully developed two-phase flow and only the test section is modeled. The test section inlet velocities are considered steady with time and are specified by the fluids actual velocities.

Mathematical formulation

For the fluid domain modeling, the commercial software FLUENT version 17.2 is employed. CFD solvers employ a set of seven equation including the conservation of mass equation in addition to the Unsteady Reynolds-Averaged Navier-Stokes (URANS) equations which are coupled and non-linear and are expressed as follows:

$$\bigtriangledown \cdot u = 0 \tag{1}$$

$$\rho \frac{D}{Dt}(u) = \bigtriangledown \cdot T - \bigtriangledown P + f_g + f_c \tag{2}$$

The terms $\bigtriangledown \cdot T$ and f_g can be written as

$$\bigtriangledown \cdot T = \bigtriangledown \cdot (\mu \bigtriangledown u) + \bigtriangledown u \cdot \bigtriangledown \mu \tag{3}$$

$$= \mu \bigtriangledown^2 u \tag{4}$$

$$f_g = \rho g \tag{5}$$

The capillary force f_c in Eq.(2) contains the source term of the continuum surface force CSF that accounts for the interface area & curvature and the surface tension [26] which is assumed to be constant in this paper. The CSF was first introduced by Brackbill et al. [27] to represent the surface tension force in each cell volume. The influence of accounting for the surface tension forces can be characterized by three dimensionless numbers that affect the flow equations. Those three numbers are Weber number "*We*", Capillary number "*Ca*" & Reynolds number "*Re*" and are expressed as follows:

$$We = \frac{\rho D_i u^2}{\sigma} \tag{6}$$

$$Re = \frac{\rho u D_i}{\mu} \tag{7}$$

$$Ca = \frac{\mu u}{\sigma} = \frac{We}{Re} \tag{8}$$

The We relates the inertia and the surface tension forces, the Re relates the inertia effects to the viscosity effects and the Ca resembles the ratio between the viscous effects to the surface tension effects. Moreover, the equations of motion are dependent on the mixture material properties that are strongly dependent on the phase volume fraction of each phase α which can be calculated for two-phase flow as follows:

$$\rho = \alpha_q \rho_q + \alpha_p \rho_p \tag{9}$$

$$\mu = \alpha_q \mu_q + \alpha_p \mu_p \tag{10}$$

Equation (11) governs the time history analysis of structure domain.

$$[M]\{\ddot{\zeta}(t)\} + [C]\{\dot{\zeta}(t)\} + [K]\{\zeta(t)\} = F(t) \tag{11}$$

The tensor of generalized structure displacement with time can be expressed as follows:

$$\zeta(t) = \zeta_x + \zeta_y + \zeta_z \tag{12}$$

$\ddot{\zeta}(t)$, $\dot{\zeta}(t)$& $\zeta(t)$ are the nodal acceleration, velocity & displacement respectively. And ζ_x, ζ_y & ζ_z are the displacements in x, y & z directions respectively. Also, Newmark time integration method was employed to solve Eq.(11)

Multiphase Model

The description of free surface can be done by two approaches, the Eulerian-Eulerian model [28–30] and the Lagrangian particle-tracking model [31, 32]. The first step in solving any multiphase problem is to determine the characteristics of the flow that need to be simulated. Gama et al. [20] stated in their work the mixture velocities, air volume fraction and the flow rate ranges used in the experiments, according to Mandhane et al. [33] flow map, most of the flow regimes belong to the slug flow regime. Similarly, the correlations developed by Beggs & Brill [34] were used and they concluded that most of the flow regimes are of the intermittent type. These correlations are used to calculate the liquid holdup

and the phases actual velocities.
There are three different Euler-Euler multi-phase models available in FLUENT:

1. The Volume of Fluid (VOF) model.
2. The Mixture Model.
3. The Eulerian Model.

These models treat the fluid phases mathematically as interpenetrating continua. The concept of volume fraction is used, where the unit cell volume can only be employed by a single phase. These volume fractions are assumed to be continuous functions of space and time and their sum is equal to one. Conservation equations for each phase are derived to obtain a set of equations, which have similar structure for all phases.
Since the phases in the present case are not interpenetrating and moving with different velocities; the VOF model [35] is the optimum choice in which the volume fractions of phases in each control volume sum to unity as follows:

$$1 = \alpha_1 + \alpha_2 + \alpha_3 + ...\alpha_q$$

The VOF scheme gives reasonable accuracy and lower computational cost compared to the other Euler-Euler multiphase models. It solves a single momentum and energy equation that is dependent on the mixture properties of the fluids. The drawback of this method is that when large slip velocity exists between the phases, the accuracy of the velocities computed near the interface can be adversely affected.
Equation (13) resembles the discretized explicit formulation of the time-dependent.

$$\frac{\alpha_q^{n+1}\rho_q^{n+1} - \alpha_q^n\rho_q^n}{\Delta t}V + \sum_f(\rho_q U_f^n \alpha_{q,f}^n) = [\sum_{p=1}^{n}(\dot{m}_{qp} - \dot{m}_{pq}) + S_{\alpha_q}]V \quad (13)$$

where $\dot{m}_{qp}$ is the mass transfer from phase q^{th} to phase p^{th}, $\dot{m}_{pq}$ is the mass transfer from phase p^{th} to phase q^{th}, $\alpha_{q.f}$ is the face value of the q^{th} volume fraction and U_f is the volume flux through the face based on normal velocity.

Turbulence Model

In some cases turbulent models fail to account for the slugs' generation specially when utilizing constant boundary conditions. In recent studies various turbulence models were being invoked for this kind of simulation.
The Standard $k-\varepsilon$ without and with scalable wall functions [36] models are the most widely used turbulence models when simulating the slug flow regime [23, 24]. However, it performs poorly for complex flow geometries that involves strong curvatures or adverse pressure gradients and separation. Instead of using the turbulent dissipation rate in the $k-\varepsilon$, the turbulence frequency rate in the standard $k-\omega$ [37, 38] with turbulence damping is utilized. This model is used for modeling complex applications due to its robustness and reduced resolution demands for an integration to the wall. It successfully predicts the turbulence scale in adverse pressure gradients. however, it has a limitation due to its strong sensitivity of the solution to free stream values for ω outside the boundary layer [39].
Czapp et al. [23] & Lu [24] suggested using the Detached Eddy Simulation (DES) & Large Eddy Simulation (LES) to improve spatial resolution and provide more accurate results, simulation of the smaller vortex structures with URANS then switching to LES or DES prior to a slug formation for this highly unsteady two-phase phenomenon. That technique would also require high computational cost compared to the other models.
Due to the limitation of the standard $k-\omega$ model in 1994, Menter et al. [40] suggested a combination between the $k-\omega$ & the $k-\varepsilon$ models near & away from the wall, respectively, leading to the SST (Shear-Stress-Transport) model that can be used to model the stratified flow regimes [41].

GRID DEPENDENCY STUDY

Three meshes were utilized for each of fluid domain and the structure domain. A combination between hexahedrons & prism elements was used for the fluid domain, and for the structure domain prism elements were used as shown in Fig. 3. The generated fluid domain mesh enhanced the mesh quality and decreased the mesh average skewness for the mesh with 1610928 elements from 0.1288 to 0.10588 compared to pure hexagonal elements. For the structure domain, the mesh with 391050 elements, the average skewness decreased from 0.35411 to 0.27904 compared to the usage of combination between hexahedrons and prism elements. Non-uniform grid refinement ratios were used as per tables 1 & 2. By optimizing the computational resources for the CFD solver most of the cases were modeled using the mesh with the moderate number of elements ”2641200 elements”.

TIME STEP DEPENDENCY STUDY

Three time steps were tested in the structural domain as shown in Tab. 3, while two time step sizes were tested for the fluid domain as per Tab. 4. The time step in the mechanical domain resembles the sampling time, when implementing the solution using time steps 0.001 & 0.0001 the discontinuity in the acceleration vs time profile decreased, but the resulted RMS values were approximately similar.

TABLE 1. GRID DEPENDENCE STUDY: FLUID DOMAIN MESH.

	Mesh 1	Mesh 2	Mesh 3
No. of Nodes	6755357	10977525	23345909
No. of Elements	1610928	2641200	5695200
Experimental Result(g/RMS) [20]	0.123	0.123	0.123
Numerical Solver Result(g/RMS)	0.1363	0.1302	0.1263
Deviation from Experiment (%)	10.81	5.853	2.682

TABLE 2. GRID DEPENDENCE STUDY: SOLID DOMAIN MESH.

	Mesh 1	Mesh 2	Mesh 3
No. of Nodes	177128	489918	1174305
No. of Elements	44240	97960	391050
Experimental Result(g/RMS) [20]	0.332	0.332	0.332
Numerical Solver Result(g/RMS)	0.3511	0.343582	0.3422
Deviation from Experiment (%)	5.753	3.489	3.072

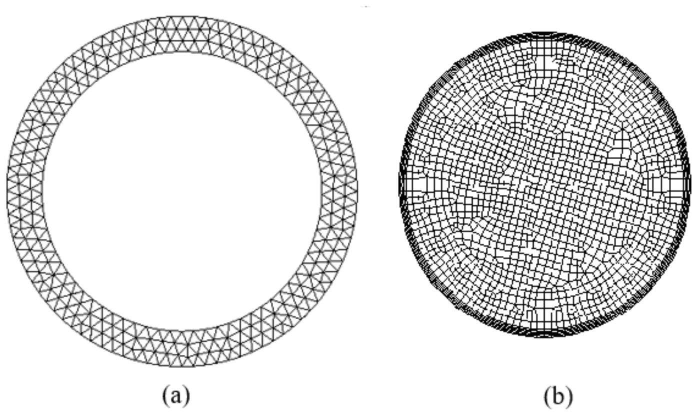

FIGURE 3. MESH PROFILES (a) STRUCTURAL DOMAIN MESHING ”391050 ELEMENTS” (b) FLUID DOMAIN MESHING ”5695200 ELEMENTS”.

CFD SOLVER RESIDUALS DEPENDENCY STUDY

Three convergence criteria were tested as shown in Tab. 5 where the deviation from the experimental results decreased slightly when increasing the orders of magnitude to reach 4.878% at 10^{-5} criterion with an error difference of 0.9757% from the 10^{-4} criterion, by optimizing the solution the case studies were modeled in this work using convergence criteria of four orders of magnitude.

TABLE 3. MECHANICAL SOLVER TIME STEP DEPENDENCE STUDY.

	Case1	Case2	Case3
Step Size (sec)	0.01	0.001	0.0001
Experimental Result (g/RMS) [20]	0.244	0.244	0.244
Numerical Solver Result (g/RMS)	0.239	0.257	0.257
Error (%)	-2.09	5.369	5.451

TABLE 4. CFD SOLVER TIME STEP DEPENDENCE STUDY.

	Case1	Case2
Step Size (sec)	0.001	0.0001
Experimental Result (g/RMS) [20]	0.123	0.123
Numerical Solver Result(g/RMS)	0.1302	0.1274
Error (%)	5.853	3.5772

TABLE 5. CFD SOLVER RESIDUALS DEPENDENCE STUDY.

	Case1	Case2	Case3
Step Size	10^{-3}	10^{-4}	10^{-5}
Experimental Result (g/RMS) [20]	0.123	0.123	0.123
Numerical Solver Result (g/RMS)	0.131	0.13	0.129
Error (%)	6.585	5.854	4.878

RESULTS AND DISCUSSION

The case of high flow velocities producing long bubbles that falls in the intermittent flow regime region was investigated, these long bubbles are called Benjamin bubbles for horizontal piping systems. The frequency response of the system strongly depends on the fluid, the structure material properties and the flow rates being introduced to the pipe.

The commonly used turbulence model for modeling slug flow is the standard $k-\varepsilon$ showed poor convergence, this is due to the instability introduced by the piping system in the vicinity of the elbow. The standard $k-\omega$ model with turbulence damping proved to enhance the solution stability and convergence. It also succeeded in predicting the flow regime transformation from stratified flow to intermittent flow.

Steady flow velocities were used as the inlet boundary conditions resulting in the production of a stratified flow regime with the liquid phase flowing in the bottom and gas phase above it due to buoyant forces, the liquid velocity then decreases due to the shear forces produced by the pipe wall and flow instabilities occurred. Waves were then generated, the amplitude of certain waves increased dramatically until they bridged the whole pipe causing the transition from stratified flow into intermittent slug flow. The periodic behavior of the slugs' generation mechanism excites the pipe in a periodic way. The measurements were taken for the first slug only. Figures 4 & 5 show the differences between the numerical and experimental frequency responses of piping system in z-direction. The proposed FSI model succeeded in predicting the RMS acceleration response with an average error of $\pm 3.8\%$.

It is noticed that for a single void fraction, as the mixture superficial velocities increases, the frequency response increases irrespective to the flow regime type.

Figure 7 shows the relationship between the liquid holdup H_L based on Brill & Beggs [34] correlations versus the superficial mixture velocity. It was noticed that for all void fractions the phases superficial velocities are inversely proportional to the liquid holdup, i.e. an increase in the injected flow rates in the pipeline results in a decrease the liquid holdup, this agrees with the work published by Tay & Thorpe [17]. Also, when the flow regime transforms from intermittent to distributed flow regime the curves' slope changes with a sudden drop in the liquid holdup. The measurement of liquid holdup can be a good indication of the void fraction of the fluid flowing through a pipeline.

At high slip velocities where the interfacial surface tension is weak the model failed to quantify correctly the flow induced vibration magnitude and the error increased dramatically, for example when the slip velocity between phases was 7.2 m/s, the acceleration response in the vertical plane reached 0.6 $g.RMS$. This value is greater than the experimental value which was supposed to be 0.1 $g.RMS$ indicating an error of 500%. Also, the

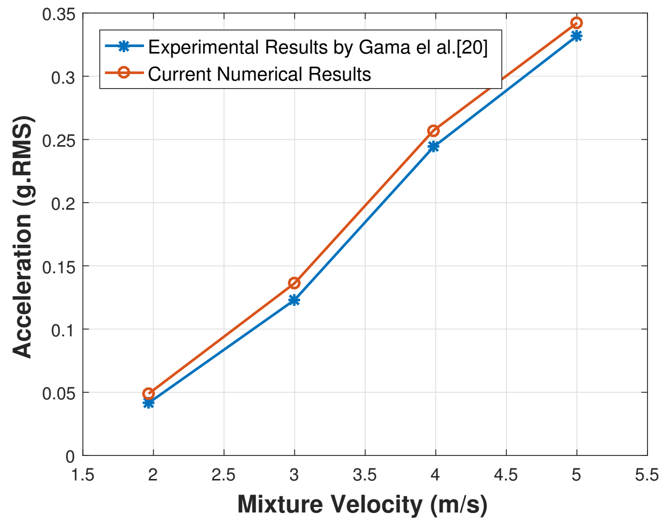

FIGURE 4. THE NUMERICAL RESULTS VS THE EXPERIMENT RESULTS FOR $\alpha_L = 0.5$

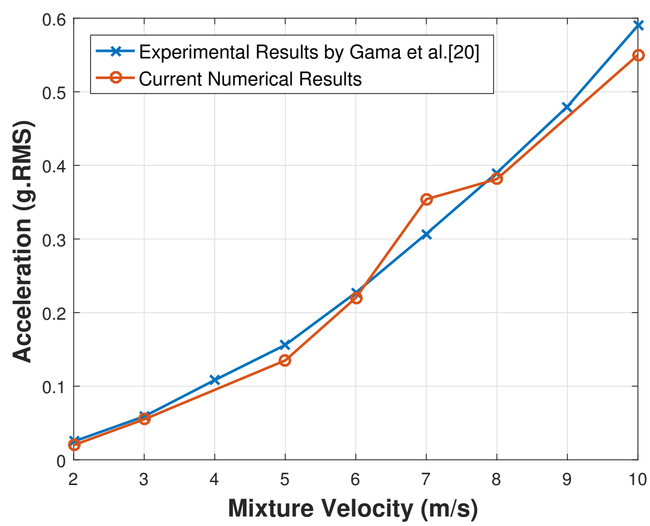

FIGURE 5. THE NUMERICAL RESULTS VS THE EXPERIMENT RESULTS FOR $\alpha_L = 0.8$

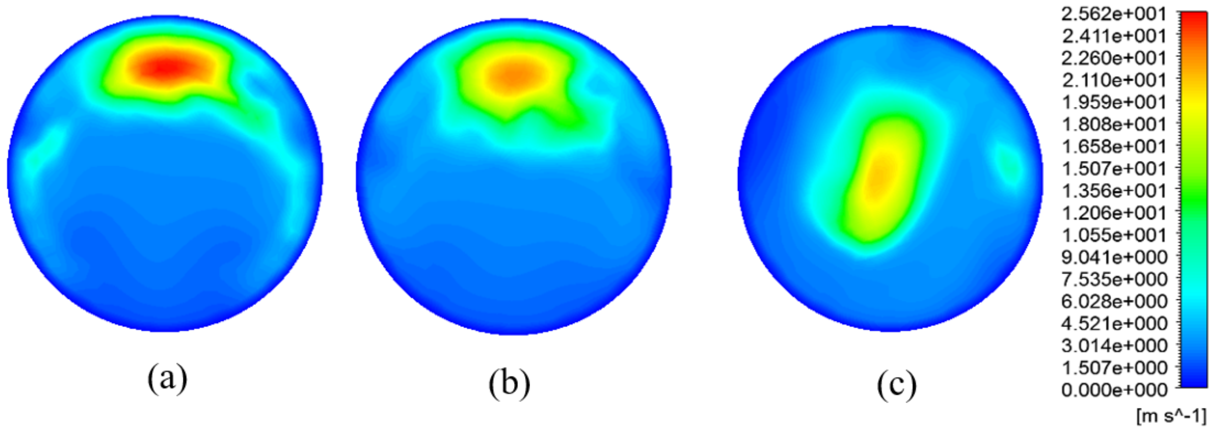

FIGURE 6. VELOCITY CONTOURS FOR GAS & LIQUID WITH INLET SLIP VELOCITIES OF 7.2 m/s. (a)VELOCITY CONTOURS AT A DISTANCE 0.45m UPSTREAM THE ELBOW MEASURED FROM THE ELBOW CENTER (b)VELOCITY CONTOURS AT A DISTANCE 0.4m UPSTREAM THE ELBOW MEASURED FROM THE ELBOW CENTER (c)VELOCITY CONTOURS AT A DISTANCE 0.25m DOWNSTREAM THE ELBOW MEASURED FROM THE ELBOW CENTER

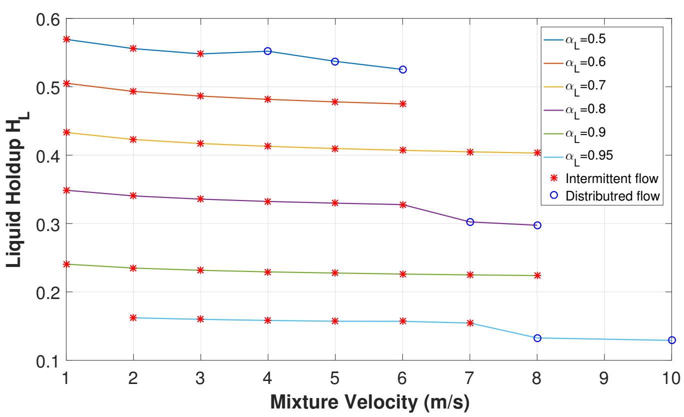

FIGURE 7. LIQUID HOLDUP VS MIXTURE SUPERFICIAL VELOCITY FOR DIFFERENT INPUT LIQUID CONTENT

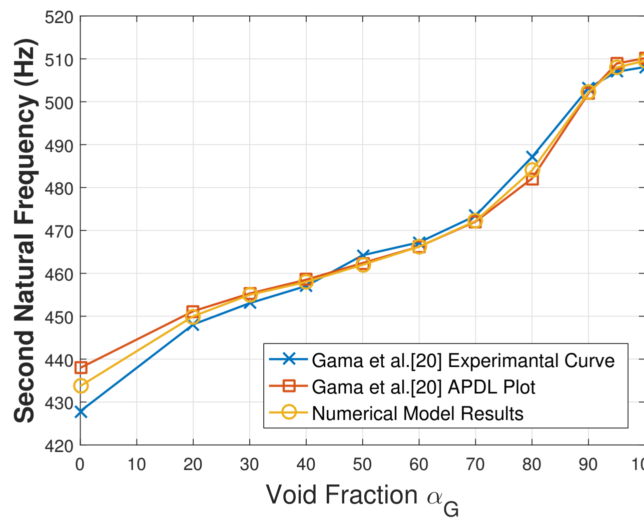

FIGURE 8. THE EFFECT OF VOID FRACTION VARIATION ON THE SYSTEM SECOND NATURAL FREQUENCY

velocity contours showed that the calculated velocities of phases in some locations reached 25.62m/s as shown in Fig.6.

These high velocities are generated due to the presence of parasitic currents [42], these parasitic currents according to Harvie et al. [43] are unphysical currents generated in the fluid region near the interface between the phases when implementing CSF. Therefore, for flows with high Ca & We the surface tension effects can be ignored and the CSF term can be canceled from Eq.(2).

The methodology proposed by Gama et al. [20] is based on the conclusion made by Evans et al. [21] for single phase flows which stated that the relationship between the pipe acceleration response and the flow rates is of deterministic type. However, to determine the flow rates in a two-phase flows the vibration response can be a good indication for the phases void fractions, Gama et al. [20] suggested that the determination of the flow void fraction can be performed by exciting a piping component with one of its natural frequencies, depending on the acceleration response one can detect the void fraction of the flow, Numerical modal analysis was performed on a model similar to the one proposed by Gama et al. [20]. The model used in the modal analysis study succeeded in predicting the pipe second natural frequency with a deviation from the experimental data of 0.42%, which is lower than the previous model proposed by Gama et al. [20], which deviated from experimental data by 0.61%, indicating an enhancement in the numerical results by 0.19%.

The physical model consisted of an acrylic pipe section of 0.5 meter long clamped from both ends, the pipe was filled with water with no net flow rate where eleven void fractions were tested (from 0% to 100%). For each of the eleven void fractions separate mesh dependence studies were performed using three meshes for each, using non uniform grid refinement ratios with minimum ratio of 1.3, the pipe was exited from 0 to 1000 *Hz*. Block Lanczos algorithm was used in the modal analysis study where the Lanczos recursion is performed. This numerical model succeeded in detecting the first six natural frequencies, it was found that the second natural frequency is the most sensitive to the variation in void fraction, this agrees with the conclusion made by Gama et al. [20], Figure 8 shows the results obtained from the proposed numerical model compared to the previous experimental and numerical data. The methodology introduced by Gama et al. [20] is based on measuring the second natural frequency, which is used to deduce the void fraction of the two-phase flow that is being conveyed, knowing both the void fraction and the vibration response of a piping element, the two-phase mixture velocity can be estimated. Accordingly the phases flow rates can be calculated using the well known two-phase equations.

The proposed numerical method succeeded in predicting the vibration response of the pipeline with an average error of $\pm 3.8\%$. This error is acceptable in some process plants, for example, in oil & gas production wells producing two phase flow. The current used techniques for measuring the production flow rate requires separation of phases which is costly in terms of initial and running costs compared to the proposed method. Additionally, some of the well-known orifice based devises that are widely used in process plants measure the flow rates with an error $\pm 5\%$.

CONCLUSION

The proposed numerical one way FSI model was verified and validated using previous experimental data and using parametric studies on the CFD and the FE codes including mesh, time step size and residuals order magnitude dependency studies. It is shown that for a certain void fraction as the mixture superficial velocity increases and the inertia effects dominate the flow. The acceleration response RMS value measured in a plane perpendicular to the pipe section increases no matter the flow pattern type. Also, for the same mixture superficial velocity the void fraction is directly proportional to the RMS value of acceleration response for the structure. The relationship between the natural frequencies and the void fraction is deterministic, for the pipe element tested in this work the second natural frequency has the greatest response to the void fraction variation compared to other natural frequencies. The void fraction can also be deduced through measuring the liquid holdup, as the superficial mixture velocity increases, the surface tension effects decreases causing a drop in the liquid holdup.

ACKNOWLEDGMENT

The authors would like to express their gratefulness to the Informatics Research Institute (IRI) in the SRTA City for supporting this research.

REFERENCES

[1] Kavanagh, W. K., Connaire, A., Payne, J., Connolly, A., McLoughlin, J., et al., 2013. "Slug response in subsea piping: Advancements in analytical methods". In Offshore Technology Conference, Offshore Technology Conference.

[2] Han, P., and Guo, L., 2015. "Numerical simulation of terrain-induced severe slugging coupled by hydrodynamic slugs in a pipeline–riser system". *International Journal of Heat and Fluid Flow,* ***56***, pp. 355–366.

[3] Irikura, M., Maekawa, M., Hosokawa, S., and Tomiyama, A., 2014. "Numerical simulation of slugging of stagnant liquid at a v-shaped elbow in a pipeline". *Applied Mathematical Modelling,* ***38***(17), pp. 4238–4248.

[4] Hill, T., Wood, D., et al., 1994. "Slug flow: Occurrence, consequences, and prediction". In University of Tulsa Centennial Petroleum Engineering Symposium, Society of Petroleum Engineers, 27960, Tusla, Oklahoma.

[5] Moros, A., and Fairhurst, P., 1999. "Production riser design: integrated approach to flow, mechanical issues, predicting riser-to-riser interaction". *Offshore International,* ***59***, pp. 88–89.

[6] Pedersen, S., Durdevic, P., and Yang, Z., 2015. "Review of slug detection, modeling and control techniques for offshore oil & gas production processes". *Ifac-papersonline,* ***48***(6), pp. 89–96.

[7] Ehinmowo, A. B., Ogunleye, O. O., and Orodu, O. D., 2016. "Experimental investigation of hydrodynamic slug mitigation potential of an intermittent absorber". *Chemical Engineering Research and Design,* ***113***, pp. 50–60.

[8] Arnold, R., Sandmeyer, D., Eickmeire, J., et al., 1972. "Production problems of a high pressure, high temperature reservoir". In Annual Technical Meeting, Petroleum Society of Canada.

[9] Marnell, D., and Winn, N., 1997. "Measurement of the forces on bends in oil pipelines". *Part II Report, Department of Chemical Engineering, University of Cambridge*.

[10] Fairhurst, C., 1983. "Two-phase transient phenomena in oil production flowlines". *BHRA the Fluid Engineering Centre (Confidential)*.

[11] Sánchez, S., Toledo, V., and Hernández, G., 1998.

"Slug flow momentum transfer analysis to determine the forces acting on a 90 elbow in a horizontal pipe". In 3rd International Conference on Multiphase Flow, Lyon, France.

[12] Tay, B. L., 2002. "Forces on pipe bends due to intermittent gas-liquid flow". PhD thesis, University of Cambridge.

[13] Tay, B. L., and Thorpe, R. B., 2014. "Hydrodynamic forces acting on pipe bends in gas–liquid slug flow". *Chemical Engineering Research and Design,* ***92***(5), pp. 812–825.

[14] Tay, B. L., and Thorpe, R. B., 2015. "Statistical analysis of the hydrodynamic forces acting on pipe bends in gas–liquid slug flow and their relation to fatigue". *Chemical Engineering Research and Design,* ***104***, pp. 457–471.

[15] Al-Hashimy, Z. I., Al-Kayiem, H. H., and Time, R. W., 2006. "Experimental investigation on the vibration induced by slug flow in horizontal pipe". *ARPN Journal of Engineering and Applied Sciences,* ***11, No.20***.

[16] Belfroid, S., Cargnelutti, M., Schiferli, W., and van Osch, M., 2010. "Forces on bends and t-joints due to multiphase flow". In ASME 2010 3rd Joint US-European Fluids Engineering Summer Meeting collocated with 8th International Conference on Nanochannels, Microchannels, and Minichannels, Vol. 3, Parts A and B, American Society of Mechanical Engineers, Montreal, Quebec, Canada, ISBN: 978-0-7918-5451-8, pp. 613–619.

[17] Tay, B., and Thorpe, R., 2004. "Effects of liquid physical properties on the forces acting on a pipe bend in gas–liquid slug flow". *Chemical engineering research and design,* ***82***(3), pp. 344–356.

[18] An, C., and Su, J., 2015. "Dynamic behavior of pipes conveying gas–liquid two-phase flow". *Nuclear Engineering and Design,* ***292***, pp. 204–212.

[19] Monette, C., and Pettigrew, M., 2004. "Fluidelastic instability of flexible tubes subjected to two-phase internal flow". *Journal of fluids and structures,* ***19***(7), pp. 943–956.

[20] Gama, A. L., dos Santos Ferreira, L. R., and Walter Filho, P. H. A., 2009. "Experimental study on the measurement of two-phase flow rate using pipe vibration". In Proceedings of 20th International Congress of Mechanical Engineering, ABCM, Gramado, RS, Brazil, pp. 15–20.

[21] Blotter, J. D., and Stephens, A. G., 126(2)(2004). "Flow rate measurements using flow-induced pipe vibration". *Fluids Engineering,* ***296(2)***, pp. 280–285.

[22] Frank, T., 2005. "Numerical simulation of slug flow regime for an air-water two-phase flow in horizontal pipes". In Proceedings of the 11th International Topical Meeting on Nuclear Reactor Thermal-Hydraulics (NURETH-11), Avignon, France, October, pp. 2–6.

[23] Czapp, M., Utschick, M., Rutzmoser, J., and Sattelmayer, T., 2012. "Investigations on slug flow in a horizontal pipe using stereoscopic particle image velocimetry and cfd simulation with volume of fluid method". In 2012 20th International Conference on Nuclear Engineering and the ASME 2012 Power Conference, Anaheim, California, USA, Vol. 3, American Society of Mechanical Engineers, pp. 477–486.

[24] Lu, M., 2015. "Experimental and computational study of two-phase slug flow". PhD thesis, Imperial College London.

[25] Li, F., Cao, J., Duan, M., An, C., Su, J., et al., 2016. "Two-phase flow induced vibration of subsea span pipeline". In The 26th International Ocean and Polar Engineering Conference, International Society of Offshore and Polar Engineers, Rhodes, Greece, ISBN:978-1-880653-88-3.

[26] Bilger, C., Aboukhedr, M., Vogiatzaki, K., and Cant, R., 2017. "Evaluation of two-phase flow solvers using level set and volume of fluid methods". *Journal of Computational Physics*.

[27] Brackbill, J., Kothe, D. B., and Zemach, C., 1992. "A continuum method for modeling surface tension". *Journal of computational physics,* ***100***(2), pp. 335–354.

[28] Spalding, D. B., 1981. "Numerical computation of multi-phase fluid flow and heat transfer". In Von Karman Inst. for Fluid Dyn. Numerical Computation of Multi-Phase Flows, Recent Advances in Numerical Methods in Fluids, pp. 139–167.

[29] Markatos, N., and Kirkcaldy, D., 1983. "Analysis and computation of three-dimensional, transient flow and combustion through granulated propellants". *International Journal of Heat and Mass Transfer,* ***26***(7), pp. 1037–1053.

[30] Markatos, N. C., 1986. "Modelling of two-phase transient flow and combustion of granular propellants". *International journal of multiphase flow,* ***12***(6), pp. 913–933.

[31] Dukowicz, J. K., 1980. "A particle-fluid numerical model for liquid sprays". *Journal of Computational Physics,* ***35***(2), pp. 229–253.

[32] Wahba, E., and Nawar, H., 2013. "Multiphase flow modeling and optimization for online wash systems of gas turbines". *Applied Mathematical Modelling,* ***37***(14), pp. 7549–7560.

[33] Mandhane, J., Gregory, G., and Aziz, K., 1974. "A flow pattern map for gasliquid flow in horizontal pipes". *International Journal of Multiphase Flow,* ***1***(4), pp. 537–553.

[34] Brill, J. P., and Beggs, H. D., 1984. *Two-phase flow in pipes*. University of Tusla, Okla.

[35] Hirt, C. W., and Nichols, B. D., 1981. "Volume of fluid (vof) method for the dynamics of free boundaries". *Journal of computational physics,* ***39***(1), pp. 201–225.

[36] Menter, F., and Esch, T., 2001. "Elements of industrial heat transfer predictions". In 16th Brazilian Congress of Mechanical Engineering (COBEM), Uberlandia, Brazil,

26-30 November, Vol. 109.
[37] Wilcox, D. C., 1988. “Reassessment of the scale-determining equation for advanced turbulence models”. *AIAA journal,* ***26***(11), pp. 1299–1310.
[38] Wilcox, D., 1991. “A half century historical review of the k-omega model”. In 29th Aerospace Sciences Meeting, Reno, NV, U.S.A, p. 615.
[39] Menter, F., 1992. “Influence of freestream values on k-omega turbulence model predictions”. *AIAA journal,* ***30***(6), pp. 1657–1659.
[40] Menter, F. R., 1994. “Two-equation eddy-viscosity turbulence models for engineering applications”. *AIAA journal,* ***32***(8), pp. 1598–1605.
[41] Vallée, C., Höhne, T., Prasser, H.-M., and Sühnel, T., 2008. “Experimental investigation and cfd simulation of horizontal stratified two-phase flow phenomena”. *Nuclear Engineering and Design,* ***238***(3), pp. 637–646.
[42] Lafaurie, B., Nardone, C., Scardovelli, R., Zaleski, S., and Zanetti, G., 1994. “Modelling merging and fragmentation in multiphase flows with surfer”. *Journal of Computational Physics,* ***113***(1), pp. 134–147.
[43] Harvie, D. J., Davidson, M., and Rudman, M., 2006. “An analysis of parasitic current generation in volume of fluid simulations”. *Applied mathematical modelling,* ***30***(10), pp. 1056–1066.
[44] Mao, Z.-S., and Dukler, A., 1989. “An experimental study of gas-liquid slug flow”. *Experiments in Fluids,* ***8***(3-4), pp. 169–182.
[45] Hoffmann, A., Hirsch, T., and Pitz-Paal, R., 2016. “Numerical investigation of severe slugging under conditions of a parabolic trough power plant with direct steam generation”. *Solar Energy,* ***133***, pp. 567–585.
[46] Launder, B. E., and Spalding, D. B., 1974. “The numerical computation of turbulent flows”. *Computer methods in applied mechanics and engineering,* ***3***(2), pp. 269–289.