Journal of Natural Gas Science and Engineering 26 (2015) 907–920

Contents lists available at ScienceDirect

Journal of Natural Gas Science and Engineering

journal homepage: www.elsevier.com/locate/jngse

Numerical analysis of quasi-steady flow characteristics in large diameter pipes with low liquid loading under high pressure

Xiaorui Guan [a], Yingliang Zhao [a], Jianjun Wang [a], Youhai Jin [a, *], Dalei Zhang [b]

[a] College of Chemical Engineering, China University of Petroleum, Qingdao 266580, PR China
[b] College of Mechanical and Electronic Engineering, China University of Petroleum, Qingdao 266580, PR China

ARTICLE INFO

Article history:
Received 8 February 2015
Received in revised form
23 July 2015
Accepted 24 July 2015
Available online 27 July 2015

Keywords:
Computational fluid dynamics
Gas–liquid flow
Film
Large diameter pipe
Low liquid loading

ABSTRACT

Computational fluid dynamics (CFD) modeling was carried out to study wavy thin film flow and droplet transport in high-pressure natural gas pipelines with low liquid loading. The interactions between droplets and thin film were considered. The results of the CFD analysis showed periodic fluctuation in the axial film distribution in the straight pipe. In the bend, gas streamlines originated from the inside of the bend. As the gas velocity was increased, more droplets were stripped, while the stripped droplets were deposited further away. Liquid film peak formation and droplet motion were attributed to the actions of centrifugal force and gravity in the bend. Secondary flow enhanced the transport of droplets to the upper side of the pipe and supplemented the upper film. The droplet fraction distribution changed periodically with time. The accumulation of droplets on the film has a periodic characteristic. Dense droplets tended to accumulate in regions of low vortices. The results of the CFD simulation were in good agreement with previous experimental data and theoretical analyses reported in the literature.

1. Introduction

Low liquid loading flow referring to flow conditions where the liquid flow rate is small compared to the gas flow rate, is widely encountered in wet gas pipelines (Banafi et al., 2014, Fadairo et al., 2014). Adequate protection of wet gas pipelines from corrosion requires knowledge of the quasi-steady flow characteristics in such pipelines (Bozzini et al., 2003). In wet gas gathering pipelines, condensates and corrosive media form a thin film on the pipe wall. The uneven distribution of the film on the pipe wall leads to localized corrosion in certain areas. The effects of mass transfer rate, wall shear stress, and electrochemical corrosion must be considered in analyses of flow-induced corrosion. The film distribution in the boundary layer has an important impact on mass transfer rate and wall shear stress, which in turn influence the progress of corrosion (Zheng et al., 2007, Parsi et al., 2014). Thin film corrosion in wet gas pipelines is different from atmospheric thin film corrosion due to reduced availability of oxygen; therefore, the effects of corrosion current and film thickness on the corrosion process may also differ between these settings. However, few studies have considered this issue. Therefore, studies of the distribution of the thin film and droplet fractions in wet gas pipelines are needed to provide a foundation for research on localized corrosion.

Computational fluid dynamics (CFD) technology is used to simulate flow patterns and other flow characteristics of various types of gas–liquid flow. CFD results and experimental data in the literature show good agreement. Ghorai and Nigam (2006) used CFD to study the influences of gas velocity, liquid volume fraction and interfacial roughness on gas–liquid flow with a volume of fluid (VOF) model. The numerical results were successfully validated against experimental data from the literature. Mito and Hanratty (2007) conducted a detailed numerical study of trajectory mechanisms for particle deposition in horizontal annular flow. Verdin et al. (2014) studied water droplet transport in a gas–liquid flow in 38-inch diameter pipes with a discrete phase model (DPM), which was validated by experimentation. The presence of secondary flow structures has been demonstrated by numerical simulations (Van't Westende et al., 2007 and Brown et al., 2009) and experimental observations (Fore and Dukler, 1995). Cadorin et al. (2010) evidenced the capability of CFD analysis to determine the energy performance of fuel transportation in a gas pipeline. CFD offers a deeper insight into the underlying physical mechanisms at work in wet gas pipelines and fosters understanding of gas–liquid flow. Currently utilized CFD models offer the capability to analyze

* Corresponding author.
E-mail address: jinyh@upc.edu.cn (Y. Jin).

http://dx.doi.org/10.1016/j.jngse.2015.07.039

some aspects of gas–liquid flow; however, they cannot be used to assess interactions between droplets and a thin film. For the simulation reported in this article, a Mixture model was coupled with the Eulerian wall film model. The droplet and thin film distributions in large-diameter pipes with low liquid loading under high pressure are assessed simultaneously in a manner barely possible using the most advanced experimental techniques currently available.

Conductance measurements are widely used in experiments to measure liquid film thickness based on the conductance between 2 probes in contact with the local liquid film. Different types of electrodes, including needle probes, parallel wire probes and flush-mounted pin probes have been used by researchers over the last decades (Koskie et al., 1989; Fossa, 1998; Li et al., 1997; Conte and Azzopardi, 2003; Belt, 2007; Geraci et al., 2007; Wang et al., 2012; Abdulkadir et al., 2014). Probes are inserted in the film to be measured, where they affect the flow state. The accuracy of conductance measurements cannot be guaranteed, especially for very thin films. Many studies concerning liquid film thickness distribution in pipes have been limited to analyzing gas–liquid flow in small-diameter pipes. For large-diameter pipes, some data on gas–liquid flow through a 180° inverted U-bend (diameter of 127 mm and a curvature ratio (R_c/D) of 3) was reported by Abdulkadir et al. (2014, 2012), who measured superficial gas velocities ranging from 3.5 to 16.1 m s^{-1} and water velocities ranging from 0.02 to 0.2 m s^{-1}. The film distribution in the 180° inverted U-bend was measured using flush-mounted pins and parallel wire probes, but such measurements may be extremely difficult to perform in pipes with lower liquid loading and larger diameter.

Experimental data is usually obtained from small pipes (no larger than 4 inches in diameter). For larger pipes, CFD is an effective method of modeling gas–liquid flow to obtain useful data. However, little data on quasi-steady flow characteristics in large-diameter pipes with low liquid loading is available in the literature. In addition, few studies have modeled gas–liquid flow with the droplets and liquid film expressed simultaneously using CFD. Here, we report the results of our CFD analysis of wavy thin film flow and droplet transport in natural gas pipelines. The interactions between droplets and thin film are considered. In addition, gas flow, film distribution, and cross-sectional droplet fraction were analyzed at 3 superficial gas Reynolds numbers (Re_{sg}).

2. CFD model

2.1. Geometric model of pipe

A geometric model of a pipe from the Puguang gas field in China is represented in Fig. 1. A localized corrosion tends to happen in this pipe. It is important to know the flow characteristics. According to in-situ measurements, the diameter of the pipe (D) is 0.4636 m, the angle of the bend is 37°47′, the curvature radius (R_c) is 2.03 m, and

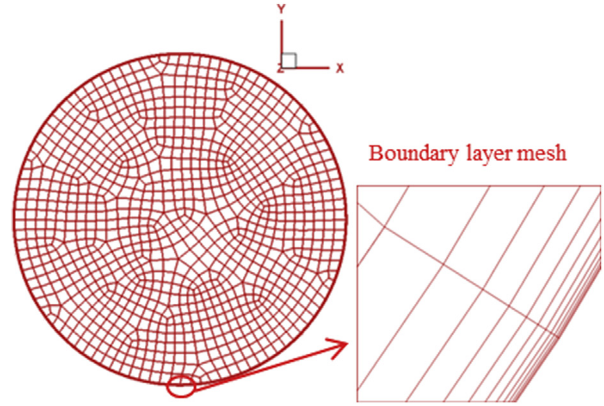

Fig. 2. Mesh of the straight pipe cross-section.

curvature ratio (R_c/D) is 4.379. The length of each horizontal and inclined section of the pipe is 9 m. The origin of the coordinates is located at the center of the outlet. The direction of gravity is negative on the Y-axis.

2.2. Meshing

A boundary layer mesh was adopted to simulate the flow and liquid film distribution near the wall of the chosen pipe. The height of the first layer of elements (d_s) was 15 μm, the expansion ratio from the first layer of elements on the surface was 1.2, and the number of layers generated from the surface was 12. The ratio of length to width for each mesh was less than 150. The total number of grid nodes was 2,792,250. The mesh of the straight pipe cross-section and bend are shown in Figs. 2 and 3, respectively. The numbers of nodes in the cross section and side wall were strictly controlled. The mesh was of high quality, with skewness less than 0.42.

2.3. Boundary conditions

2.3.1. Inlet conditions

The flow data for the gas–liquid simulation (based on data from a separator in the Puguang gas field) is provided in Table 1. The properties of the natural gas in the pipeline were converted for the simulation as follows: pseudo-critical parameters were corrected by the Wichest-Aziz method, the deviation coefficient was checked using a Standing-Katz graph, and the compressibility factor (0.86) of the natural gas was calculated. The gas flow rate in-situ was calculated. The gas density in-situ (ρ_g) was calculated according to the law of conservation of mass. The gas viscosity in-situ was corrected for the effects of temperature and pressure. The parameters needed for the simulation were calculated using Eqs. (1)–(3) (Pope, 2000):

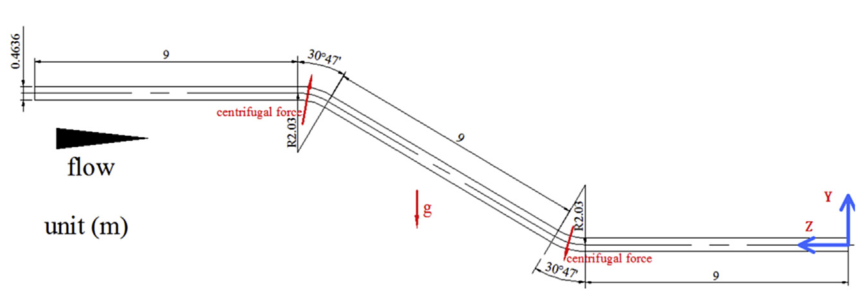

Fig. 1. Schematic diagram of the pipe.