Journal of Petroleum Science and Engineering 146 (2016) 134–141

Contents lists available at ScienceDirect

Journal of Petroleum Science and Engineering

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

Modeling by computational fluid dynamics simulation of pipeline corrosion in CO_2-containing oil-water two phase flow

Haitao Hu, Y. Frank Cheng*

Department of Mechanical & Manufacturing Engineering, University of Calgary, Calgary, AB, Canada T2N 1N4

A R T I C L E I N F O

Article history:
Received 26 February 2016
Received in revised form
8 April 2016
Accepted 25 April 2016
Available online 26 April 2016

Keywords:
Pipelines
Internal corrosion
Oil-water two phase fluid flow
Wall shear stress
Computational fluid dynamics
Corrosion modeling

A B S T R A C T

In this work, computational fluid dynamics (CFD) simulations were performed to determine the oil/water volume fraction distribution and wall shear stress in oil-water two phase fluid flow in pipelines. A CFD based empirical model was developed to predict the corrosion rate of pipelines in CO_2-containing oil-water two phase flow. The flow pattern simulation is able to locate potential corrosion occurrence in pipelines by identifying the sites for water accumulation. For horizontal pipes, at low flow velocities, i.e., 0.2, 0.3 and 0.5 m/s, the water phase accumulates at the bottom of the pipe. For upward inclined pipelines, at low flow velocities, the water phase accumulates at the bottom of the pipe, especially the upward part of the inclined pipe, even in fluids containing a very small amount of water, such as 5% water. For downward inclined pipelines, at low flow velocities, both the straight and the downward parts of the pipe around the elbow are the locations to accumulate water, presenting the likely area for corrosion to occur. The water-wetting of the pipe wall can be improved by increasing the flow velocity. The maximum wall shear stress is observed at the top of the elbow and the bottom of the upper part of the upward inclined pipe, where an accelerated corrosion would be experienced. For the downward inclined pipe, the bottom of the elbow and the top of the downward part of the pipe experience the highest wall shear stress. The bottom of the elbow may suffer from enhanced corrosion. The CFD based empirical model is able to predict corrosion rate of pipelines, with the modeling results validated by actual measurements.

1. Introduction

Both mechanistic and kinetics aspects of internal corrosion of pipelines in petroleum production have been investigated (Ikeda et al., 1984; Kermani and Morshed, 2003; Zhang and Cheng, 2011). A wide variety of models have been proposed to predict the corrosion rate of pipelines in CO_2-containing aqueous solutions (Kanwar and Jepson, 1994; Dayalan et al., 1995; Li et al., 2009; Cui et al., 2013).

An accurate modeling and prediction of pipeline corrosion is critical for industry to manage the asset integrity. The models available in numerous publications can be improved in at least two aspects. First, the fluid carried by upstream pipelines is not a single phase aqueous solution. Instead, the fluid contains at least two phases, i.e., oil and water, with corrosive gases such as CO_2 dissolved in the water. Any model applying for steel corrosion in single phase aqueous solutions is not able to describe the reality of internal corrosion of pipelines. The modeling results do not reproduce the corrosion process occurring internally, and they do not provide an accurate prediction of corrosion rate of the steel either. However, most of published models for pipeline internal corrosion apply for aqueous environments only (Turgoose et al., 1990; Nesic, 2007). Second, in fluid flow, water and oil can be either mixed to form oil-water emulsions or separated with both phases flowing as dispersed. The fluid flow pattern, which is dependent on fluid hydrodynamics, fluid properties, and pipe geometry and inclination, can cause the pipe steel wetted by either oil or water (Da Silva et al., 2006). When the steel is oil-wetted, corrosion would not occur. Internal corrosion is possible only when the pipe steel is wetted by water. Determination of the locations where water-wetting occurs and the parametric effects on the water-wetting of the pipe wall is critical to predict the location for internal corrosion to occur in oil-water two phase flow in pipelines.

The pipeline internal corrosion is under a synergistic effect of electrochemical reactions, mass transfer and wall shear stress exerted by fluid flow. It was confirmed that the CO_2 corrosion of steels is directly related to the multiphase fluid flow, and the proposed model for corrosion rate prediction should include the effect of fluid mechanics (Jepson et al., 1996; Nesic et al., 2004;

* Corresponding author.
E-mail address: fcheng@ucalgary.ca (Y.F. Cheng).

http://dx.doi.org/10.1016/j.petrol.2016.04.030
0920-4105/

Zhang and Cheng, 2010). Efird et al. (1993) work found that an enhanced wall shear stress due to fluid flow could accelerate corrosion of pipe steels. The influence of fluid flow on steel corrosion that occurs in a wide variety of dynamic apparatuses was studied, and the results showed that the fluid flow velocity and flow pattern affect greatly corrosion rate of the steel (Zeisel and Durst, 1990).

In this work, computational fluid dynamics (CFD) simulations were performed to determine the distribution of oil/water volume fractions and wall shear stress in oil-water two phase fluid flow in pipelines. An empirical model was developed to predict the corrosion rate of pipelines in CO_2-containing oil-water two phase flow by integrating the CFD simulation. The modeling was conducted in fluids containing two types of oil, and the results were validated by actually measured corrosion rate.

2. Methodology

2.1. Operating parameters

The fluid flow in pipelines was oil-water two phase flow, and CO_2 is dissolved in water. The water is not at CO_2 saturation. In reality, the content of CO_2 in the fluid carried by pipelines is not sufficient to reach the saturation. The pipe is either horizontal or inclined, with a diameter of 101.6 mm. The Canadian crude oil with the properties listed in Table 1 was used for simulation. The fluid flow velocities of 0.2, 0.3, 0.5 and 1 m/s were considered. The water/oil weight percentages, i.e., 95% water −5% oil, 80% water-20% oil, 60% water-40% oil, 40% water-60% oil, 20% water-80% oil and 5% water-95% oil, were included. To simulate the brine properties in the fluid, 0.02 M NaCl and 0.02 M $NaHCO_3$ solutions were added (Zhang and Cheng, 2009a). The temperature was set at 40 °C, and the operating pressure was set as atmospheric pressure. The pipe in simulation was 5 m in length. For pipeline inclinations, the horizontal and inclined parts of the pipe were 2.5 m in length, and the inclination angle, either upward or downward, was 30°.

2.2. Meshing and modeling

The pipe geometry and grid generation was performed using a GAMBIT software, where the hexahedral mesh was chosen due to its capability to provide a high-quality solution with few number of cells (Zeisel and Durst, 1990). The meshed model used in this work is shown in Fig. 1.

2.3. Assumptions and initial conditions

The fluid flow was considered as transient, and the effect of temperature fluctuations on the fluid flow was negligible. Phase changes were not considered in this work. There was no mass transfer between the phases (Desamala et al., 2014).

The CFD simulation and analysis was conducted using Fluent 15.0 software, where the realizable *k*-epsilon turbulent model was used. While standard wall functions were selected, it was assumed that the fluid was stagnant at the wall of the pipe (Hancock and Bush, 2002). At the pipe inlet, the velocity inlet boundary condition was used to define the actual velocity of the phases. At the outlet of the pipe, the pressure outlet boundary condition was used. The wall boundary condition was used to bind the fluid and the solid regions.

Under-relaxation factors, including momentum, volume fraction, turbulent kinetic energy and turbulent dissipation rate, were set as 0.3. A second order upwind discretization scheme was used for calculations of momentum equation, volume fraction, turbulent kinetic energy and turbulent dissipation rate. The convergence criterion was based on the residual value of the calculated variables including mass, velocity components, turbulent kinetic energy, turbulent dissipation rate and volume fraction. The simulation was time dependent (transient) with 200 time steps, 0.05 time step size and 100 iterations at each time step size (Alias et al., 2014).

Table 1
Properties of the oil contained in the fluid for CFD simulation.

Canadian bow river heavy crude oils	
Density (kg/m^3)	914
API gravity (°)	23
Interfacial tension (N/m)	0.032
Vapour pressure (kPa)	64.3
Pour point (°C)	< −30
Viscosity at 10 °C (cSt)	184
Viscosity at 20 °C (cSt)	99.7
Viscosity at 30 °C (cSt)	59.0
Viscosity at 40 °C (cSt)	37.2
Viscosity at 45 °C (cSt)	30.2

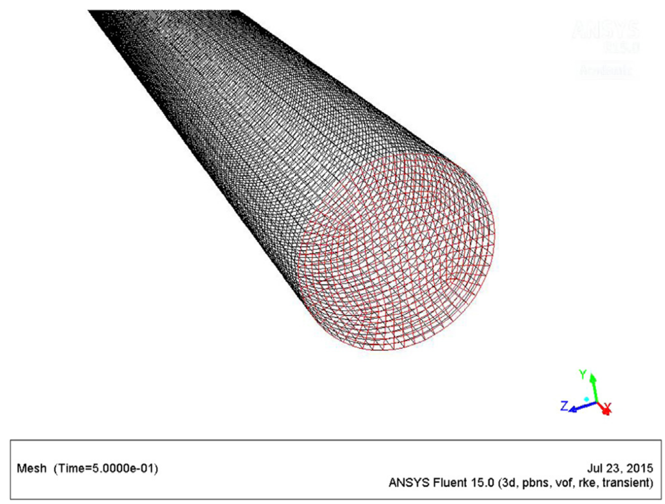

Fig. 1. Meshing of model used in CFD simulation.

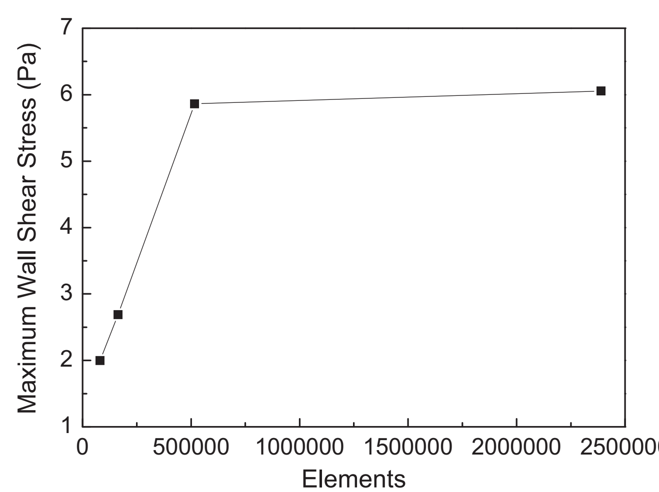

Fig. 2. Dependence of the maximum wall shear stress exerted on the pipe on the grid of elements in the pipe carrying 60% oil-40% water at a flow velocity of 0.2 m/s.