Powder Technology 275 (2015) 12–24

Contents lists available at ScienceDirect

Powder Technology

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

Numerical investigation of the process and flow erosion of flushing oil tank with nitrogen

Hongjun Zhu *, Qinghua Han, Jian Wang, Siyuan He, Dong Wang

State Key Laboratory of Oil and Gas Reservoir Geology and Exploitation, Southwest Petroleum University, Chengdu, Sichuan 610500, China

A R T I C L E I N F O

Article history:
Received 14 September 2014
Received in revised form 31 December 2014
Accepted 27 January 2015
Available online 3 February 2015

Keywords:
Flow erosion
CFD
Oil tank
DPM
VOF

A B S T R A C T

A computational fluid dynamic (CFD) model coupling with volume of fluid (VOF) method and discrete phase model (DPM) has been used to predict the flow erosion rate in an oil tank and blowdown pipe during the process of gas flushing. An initialization method by injecting particles from a tank wall is developed to construct the initialization distribution of residue particles. The flow field distribution of oil–gas–water flow and the erosion rate on the wall surface during emptying process can be captured under different operating conditions with different fluid parameters. The applicability of the approach is verified by comparing the simulated results with the measurements. The effects of fluid parameters such as gas inlet rate, oil density and viscosity, as well as the operating parameters such as residual oil level and residue particle diameter, are evaluated. In general, the required flushing time and erosion rate are all sensitive to fluid parameter changes and operating condition changes. It is found that the flow erosion is related to the emptying process of oil, which has a sharp rise in the end of the process. High inlet speed, large oil density, small oil viscosity, high oil level and large particle diameter can result in severe erosion.

1. Introduction

In the oil processing and storage, the separator and oil tank are the key equipments, which frequently require cleaning after a specified interval of time. Because of the structural complexity and to meet safe and efficient cleaning, inert gas (nitrogen) flushing is a common method used in actual operations. However, residual oil and residue particles, such as sand and catalyst, usually remain in the bottom of the tank or accumulate at the tank wall. In the flushing process, residue particles carried by fluid may result in undesired superficial erosion on the tank and blowdown pipe wall. If an eroded hole appears, some catastrophic consequences, such as oil spill, as well as considerable economic loss and liability to the oil company [1], may be brought about. Therefore, it is imperative to find the severe erosion spots in the flushing process and predict the erosion rate to estimate the service life of the tank or blowdown pipe.

Flow erosion, as a hot issue in industrial operations, has attracted many investigators to conduct physical or numerical modeling [2–5]. Since the early 1990s, with the purpose of saving time and resources and avoiding potential risks, computational fluid dynamics (CFD) as a reliable tool has been widely used for particle erosion prediction in pipe bends, elbows, ducts, tees and related geometries [6–11]. Numerical simulations have been conducted by Gabriel et al. [7] to predict the erosion due to particles in an elbow pipe with a 90° curvature angle. They found that the Oka model can give relatively accurate predictions. Stack and Abdelrahman [3] have evaluated the effects of particle concentration on the erosion of the inner surfaces of a circular pipe with a 90° bend using a commercial CFD code. Derrick and Michael [6] also used a CFD code to model particle-laden flow and predict erosion in four different 90° square cross-section bends. Zhang et al. [4] have investigated the effects of slurry velocity, bend orientation and angle of the elbow on the location of maximum erosive wear damage by CFD combined with discrete element method (DEM). Tan et al. [9] also used a CFD and DEM model to capture the key features of solid–fluid multi-phase flow and predict the location of maximum erosive wear damage. In our previous work, effects of operation, structure and fluid parameters on erosion of needle valve [8] and erosion of drill pipe [10] due to particle-laden gas flow have been studied by a CFD and DPM model. The flow erosion rates were also predicted by an erosion model. More recently, Zeng et al. [2] have performed an experimental and numerical investigation on erosion behavior of X65 pipeline elbow. The erosion rates at different locations of the elbow were quantified by using array electrodes. And the numerical results of the erosion rate were in good accordance with the test. It was found that change in hydrodynamics is the main reason for the variation of the erosion rate at different locations. Besides, anti-erosion methods were proposed and studied by several investigators. Chen et al. [5] have carried out both physical and numerical modeling of the relative erosion severity between plugged tees and elbows in dilute gas/solid two-phase flow. The erosion effects in three 90° duct bend gas–solid flows with different ribs are numerically evaluated by Fan et al. [11].

* Corresponding author. Tel.: +86 28 83032206.
E-mail address: ticky863@126.com (H. Zhu).

http://dx.doi.org/10.1016/j.powtec.2015.01.062
0032-5910/

Erosion caused by gas–solid or liquid–solid two-phase flow is the concern of these studies. Few researchers have paid attention to particle erosion resulting from gas–liquid–solid three-phase flow. However, there are residual oil remains in the bottom of the tank, resulting in gas–liquid two-phase for continuous phase. Whether flow erosion is related to the emptying process of residual oil, is a key problem for estimating the service life of the tank and pipe systems. Moreover, most previous studies focus on the flow erosion of steady flow, while it is an apparent unstable flow for gas flushing. Furthermore, in the above simulation researches, particles as discrete phase are injected at the same inlet boundary with fluid flow. However, the residue particles that accumulated at the tank wall are not carried by nitrogen from the gas inlet. So a new initialization method is required to form the residue particles before the simulation of gas flushing process.

In this study, the volume of fluid (VOF) method coupling with discrete phase model (DPM), as a Eulerian–Lagrangian approach, is used to capture the continuous phase flow and discrete particles, respectively. And an initialization method by injecting particles from the tank wall is developed to construct the initialization distribution of residue particles. The flow field distribution in the process of gas flushing and the flow erosion varied with the emptying process of residual oil are obtained by a series of non-steady simulations. The effects of gas inlet rate, residual oil density, residual oil viscosity, residual oil level, and residue particle diameter are discussed successively.

2. Problem description

In practice, oil tanks have a variety of sizes. The heights of the vast majority of tanks are more than 10 m. In order to facilitate experiments, the experimental tank size is a reduced scale of an actual one. And the numerical simulation of the tank in the current study is conducted in the same geometry size as the test for comparison. Fig. 1 shows the diagram of the computational domain, and the detailed sizes are marked on the figure. The internal space of the oil tank is composed of a cylinder and a hemisphere, which is a rotation axisymmetric construction. So a two-dimensional geometric model is established.

At the bottom of the oil tank, a blowdown branch composed of two bends connects the tank to the blowdown main pipe. The lengths of vertical straight pipe sections connecting the tank and blowdown main pipe are 0.2 m and 0.5 m, respectively. And the curvature and diameter ratios of two elbows are both 3. Connecting the two elbows is a horizontal tube with a length of 0.2 m. The length of the horizontal blowdown main pipe is 4 m. And the connection point of the branch pipe and main pipe just sits at the middle of the main pipe.

Velocity inlet boundary conditions are used for the gas inlet at the top of the tank and water inlet at the left of the blowdown main pipe. The size of the gas inlet setting in the center of the tank top is 0.05 m. Outflow boundary condition is employed for the outlet of computational domain (the right of the blowdown main pipe). The gas inlet velocity is set to 0.5 m/s, 1.0 m/s, 1.5 m/s, 2.0 m/s or 2.5 m/s, in order to observe the effect of inlet rate. However, the water inlet velocity is fixed at 2.0 m/s and the pressure at water inlet is fixed at 0.5 MPa, in order to facilitate comparative analysis. No slip boundary conditions are assumed for both the tank and pipe wall.

As shown in Fig. 1, the residual oil level is calculated from the top wall of the main pipe, which is a variable in different simulation cases in order to observe the effect of liquid level. The height of residual oil is usually no more than 40% of the height of an oil tank. Therefore, five different residual oil levels, 1.1 m, 1.2 m, 1.3 m, 1.4 m and 1.5 m, are selected in the simulations. Twenty-one cases listed in Table 1 are modeled and analyzed in this paper to compare the different influences among different gas inlet rates, oil densities, oil viscosities, residual oil levels and residual particle diameters. Besides liquid level, gas inlet rate, oil density, oil viscosity and particle diameter are chosen as variables. In practice, the gas inlet rate ranges from 0.5 m/s to 3.0 m/s. For analyzing the effect of the inlet rate, five rates, 0.5 m/s, 1.0 m/s, 1.5 m/s, 2.0 m/s and 2.5 m/s, are adopted. The range of oil density is from 700 kg/m^3 to 980 kg/m^3, in which 700 kg/m^3, 750 kg/m^3, 800 kg/m^3, 850 kg/m^3 and 900 kg/m^3 are used in this study. Oil viscosity

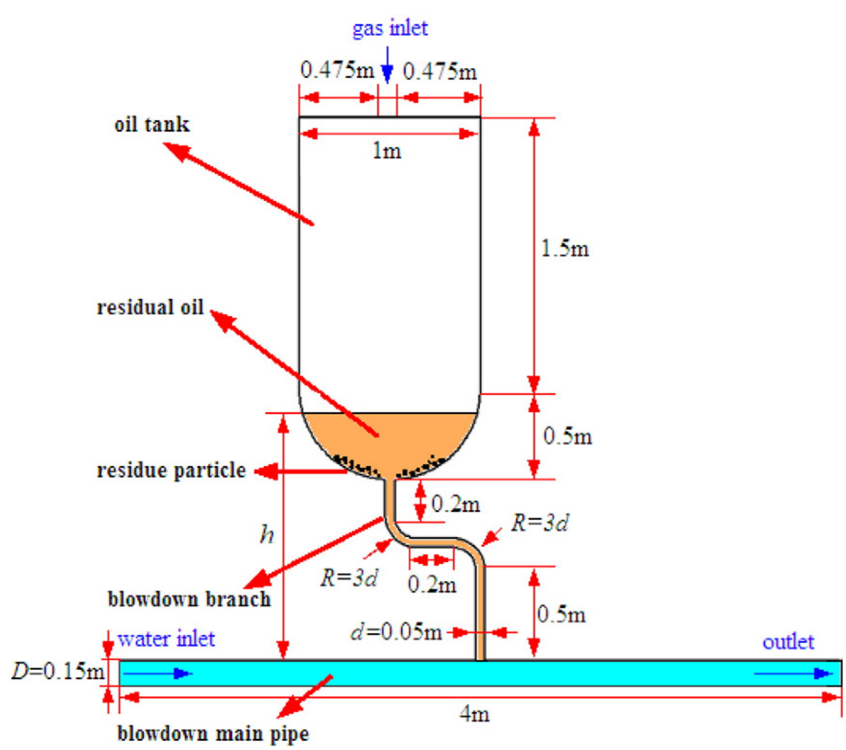

Fig. 1. Schematic diagram of oil tank and boundary conditions.