ACS OMEGA

Open Access

http://pubs.acs.org/journal/acsodf

Article

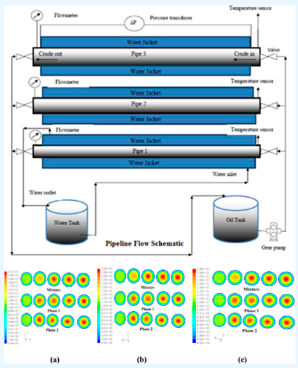
Two-Phase Crude Oil−Water Flow Through Different Pipes: An Experimental Investigation Coupled with Computational Fluid Dynamics Approach

Shirsendu Banerjee,* Anirban Banik, Vinay Kumar Rajak, Tarun Kanti Bandyopadhyay, Jayato Nayak, Michał Jasinski, Ramesh Kumar, Byong-Hun Jeon,* Masoom Raza Siddiqui, Moonis Ali Khan, Sankha Chakrabortty, and Suraj K. Tripathy*

Cite This: *ACS Omega* 2024, 9, 11181−11193

Read Online

ABSTRACT: The present study deals with two-phase non-Newtonian pseudoplastic crude oil and water flow inside horizontal pipes simulated by ANSYS. The study helps predict velocity and velocity profiles, as well as pressure drop during two-phase crude-oil−water flow, without complex calculations. Computational fluid dynamics (CFD) analysis will be very important in reducing the experimental cost and the effort of data acquisition. Three independent horizontal stainless steel pipes (SS-304) with inner diameters of 1 in., 1.5 in., and 2 in. were used to circulate crude oil with 5, 10, and 15% v/v water for simulation purposes. The entire length of the pipes, along with their surfaces, were insulated to reduce heat loss. A grid size of 221,365 was selected as the optimal grid. Two-phase flow phenomena, pressure drop calculations, shear stress on the walls, along with the rate of shear strain, and phase analysis were studied. Moreover, velocity changes from the wall to the center, causing a velocity gradient and shear strain rate, but at the center, no velocity variation (velocity gradient) was observed between the layers of the fluid. The precision of the simulation was investigated using three error parameters, such as mean square error, Nash-Sutcliffe efficiency, and RMSE-standard deviation of observation ratio. From the simulation, it was found that CFD analysis holds good agreement with experimental results. The uncertainty analysis demonstrated that our CFD model is helpful in predicting the rheological parameters very accurately. The study aids in identifying and predicting fluid flow phenomena inside horizontal straight pipes in a very effective way.



1. INTRODUCTION

Transporting crude oil from remote sources to refineries poses significant challenges, primarily when dealing with heavy crude oil. This transportation relies on pipelines with powerful pumps, but it encounters issues such as pressure loss and friction-induced deceleration or acceleration.[1] To mitigate these challenges, a common approach involves mixing the heavy crude oil with water, which reduces viscosity and aids in transportation. However, this introduces complexity through multiphase oil−water flow, where various parameters like the velocity of the mixture, transport pipe diameter, temperature, volume fraction, and pressure significantly impact the flow behavior. Furthermore, the substantial viscosity difference between crude oil and water complicates this process, necessitating careful engineering considerations for efficient and safe transportation of crude oil over long distances from its remote sources to processing facilities.[2]

Computational modeling and analysis of two-phase crude oil−water systems may enhance our present knowledge related to the transport mechanisms of oil−water mixtures in pipelines. The mainstream experimental and research work on two-phase oil−water flow focuses on flow pattern identifications such as intermittent, stratified, dispersed, core-annular, and the amalgamation of all the above.[3,4] However, computational modeling and analysis, like 3D computational fluid dynamics (CFD), have grown over the years as very important simulation tools due to their flexibility and cost-effectiveness for in-house studies.[5] The application of the CFD-based solver ANSYS for single and two-phase crude oil flow through pipes has been utilized by Kumar et al.,[6] Parvini et al.,[7] and Walvekar et al.[8] Pouraria et al.[9] applied CFD-based models for determining the flow patterns of oil−water. They utilize a general k-epsilon turbulence model in conjunction with the Eulerian−Eulerian method. The numerical results acquired were juxtaposed with experimental data found in the literature, focusing on either the in situ Sauter mean diameter or water volume fraction. The comparison between the

Received: July 21, 2023
Revised: January 23, 2024
Accepted: February 9, 2024
Published: March 1, 2024

https://doi.org/10.1021/acsomega.3c05290
ACS Omega 2024, 9, 11181−11193