Advances in Engineering Software 86 (2015) 49–54

Contents lists available at ScienceDirect

Advances in Engineering Software

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

Integration of computational fluid dynamics simulation and statistical factorial experimental design of thick-wall crude oil pipeline with heat loss

Wanwisa Rukthong [a], Wichapun Weerapakkaroon [b], Ungkana Wongsiriwan [b], Pornpote Piumsomboon [a,c], Benjapon Chalermsinsuwan [a,c,*]

[a] Fuels Research Center, Department of Chemical Technology, Faculty of Science, Chulalongkorn University, 254 Phayathai Road, Patumwan, Bangkok 10330, Thailand
[b] PTT Research & Technology Institute, PTT Public Company Limited, 555 Vibhavadi Rangsit Road, Chatuchak, Bangkok 10900, Thailand
[c] Center of Excellence on Petrochemical and Materials Technology, Chulalongkorn University, 254 Phayathai Road, Patumwan, Bangkok 10330, Thailand

ARTICLE INFO

Article history:
Received 5 November 2014
Received in revised form 31 March 2015
Accepted 12 April 2015
Available online 22 April 2015

Keywords:
Thick-wall
Pipeline
Crude oil
Wax appearance distance
Experimental design
Computational fluid dynamics

ABSTRACT

The aim of this study was to explore the heat transfer behavior between convection and conduction in the thick wall crude oil pipeline with laminar unsteady state flow using integration of developed computational fluid dynamics model and statistical experimental design. The governing equations were employed to investigate the effects of wall thickness, wall thermal conductivity, surrounding heat transfer coefficient and ambient temperature on transport profile using statistical experimental design and to locate an origin point where wax precipitate in the pipeline (wax appearance distance) by using response surface methodology (RSM). A good agreement between the model and literature experimental data suggests that the proposed numerical scheme is suitable for simulating the transport profile in pipeline and predicting the phenomena for any other conditions. From the statistical analysis, it was found that, surrounding heat transfer coefficient and ambient temperature were the major effect parameters on the wax appearance distance.

1. Introduction

Transportation of crude oil by pipelines is the general process in petroleum industry. Generally, pipelines are made of various materials which have different ability to conduct heat. Consequently, the heat affects the transport properties of the crude oil leading to operational efficiency. The proper selection of materials for the pipeline brings the effective production system [1]. Stainless steel has the thermal conductivity of 15.1 W/m K while copper has the very high thermal conductivity of 401.0 W/m K [2]. This variety of thermal conductivity affects the performance of a heat exchanger between crude oil and surrounding outside the pipe [3]. In this study, the conjugated heat transfer coupling between wall convection and conduction were concerned [4]. The analysis of this problem was carried out to study the effect of parameters which related to thick-wall problem such as wall thickness, pipe thermal conductivity, surrounding heat transfer coefficient and ambient temperature on the hydrodynamics changing in transport profile inside the pipeline. All these parameters have an important role in designing the pipeline system.

The heat and mass transfers of crude oil through pipelines are very complicated process because there are many factors which affect transport profile [5]. The knowledge in computational fluid dynamics (CFD) is therefore required for better understanding the flow behavior and for predicting the phenomena inside pipeline. Wang et al. [6] determined an optimized pigging frequency using experimental method and wax deposition model. Zhu et al. [7] investigated the thermal influential factors affecting the crude oil temperature in double pipelines system using the computational fluid dynamics methodology. However, in most of previous studies focusing on convective heat transfer, wall conduction was ignored because of the assumption of extremely thin wall. This may distort crude oil transport profile at the wall boundary condition. Thus, besides the heat convection, the heat conduction through the pipe wall should also be considered.

The aim of this study was to develop the computational fluid dynamics model for investigating the effect of thick-wall pipeline on the transport profile using statistical experimental design. In the literature, the experimental results on integration of

* Corresponding author at: Fuels Research Center, Department of Chemical Technology, Faculty of Science, Chulalongkorn University, 254 Phayathai Road, Patumwan, Bangkok 10330, Thailand.

E-mail address: benjapon.c@chula.ac.th (B. Chalermsinsuwan).

http://dx.doi.org/10.1016/j.advengsoft.2015.04.006
0965-9978/

Nomenclature

Symbols

A	wall thickness/coded factor of wall thickness (–)
B	pipe thermal conductivity/coded factor of pipe thermal conductivity (–)
C	surrounding heat transfer coefficient/coded factor of surrounding heat transfer coefficient (–)
D	ambient temperature/coded factor of ambient temperature (–)
D_{pipe}	pipe diameter (m)
h	heat transfer coefficient (kW/m^2 °C)
h_i	inner wall heat transfer coefficient (kW/m^2 °C)
h_o	outer wall heat transfer coefficient (kW/m^2 °C)
k_p	thermal conductivity (kW/m °C)
L	pipe length (m)
r_i	inner wall radius (m)
r_o	outer wall radius (m)
R	heat resistance (m^2 °C/kW)
S_ϕ	source term
t	time (s)
T	temperature (°C)
u	x-velocity (m/s)
v	y-velocity (m/s)
x	x-direction
Δx	wall thickness (m)
y	y-direction
z	wax appearance distance (m)

Greek symbols

ϕ	dependent variable
ρ	density (kg/m^3)
Γ	diffusion coefficient

Subscript

a	ambient
f	fluid/crude oil

computational fluid dynamics simulation and statistical experimental design were limited. This analysis will be helpful for preventing and solving real problems which may emerge during the crude oil transportation such as precipitation of wax.

2. Methodology

2.1. Computational fluid dynamics model

The fluid flow problem can be solved by using computational fluid dynamics (CFD) technique. CFD is a very powerful tool for analyzing the system involving fluid flow, heat transfer, mass transfer and other special phenomena such as chemical reaction [8,9] by solving the mathematical equations which govern these processes using numerical methods. To develop the CFD model, governing equations which are in partial differential form are transformed to algebraic form using finite volume numerical solution techniques [10]. Crude oil was considered as incompressible fluids. The generalized conservation equation for two-dimensional flow system where the dependent variable is denoted by ϕ can be written as:

$$\frac{\partial(\rho\phi)}{\partial t}+\frac{\partial(\rho u\phi)}{\partial x}+\frac{\partial(\rho v\phi)}{\partial y}=\frac{\partial}{\partial x}\left(\Gamma\frac{\partial\phi}{\partial x}\right)+\frac{\partial}{\partial x}\left(\Gamma\frac{\partial\phi}{\partial y}\right)+S_\phi \quad (1)$$

The two-dimensional Cartesian co-ordinates were employed as an approximation for the cylindrical pipeline system. In Eq. (1), there are the transient and the convective terms in x and y directions on the left hand side and the conductive term in x and y directions and the source term on the right hand side. Eq. (1) will turn into the continuity equation by replacing ϕ with 1. In addition, when ϕ is substituted by u, v and T, Eq. (1) will become the conservation of momentum in x and y directions and the conservation of energy, respectively.

2.2. Boundary conditions and properties

The problem was analyzed to investigate the heat exchange among surrounding, pipe wall and crude oil [11]. The schematic drawing of the thick-wall pipe problem is shown in Fig. 1. This crude oil pipeline was chosen from the real crude oil pipeline in the literature information. The flow had Peclet number of 16,300 and Biot number of 0.07–22.00 depending on the condition. Because the Peclet number is high, convection is important to flow in this system. In addition, this study Biot numbers vary from the condition that conduction inside the pipeline is unimportant to important. Heat flows from crude oil to the surrounding through a two-dimensional Cartesian thick-wall pipe with conductive heat coefficient of k_p in the wall and convective heat coefficient of h_i and h_o at the inner and outer surfaces of the wall pipe, respectively. Inner wall convection coefficient is determined from Nusselt number. The Nusselt number in laminar flow of circular tube is 3.66 [2]. The heat resistance (R) through a thick-wall pipe is

$$R=(1/h_i)+(\Delta x/k_p)+(1/h_o) \quad (2)$$

The wall thickness is the difference between the radiuses of outer and inner walls ($\Delta x = r_o - r_i$). The formula was added as an additional source term in the energy conservation equation at the wall boundary condition.

2.3. Numerical solution

To compute the solution of governing equations for each control volume on the staggered grid arrangement, SIMPLE (Semi Implicit Method Pressure-Linked Equation) algorithm was employed [9]. First order upwind differencing scheme and Tri-Diagonal Matrix Algorithm (TDMA) were applied to solve the interface problem and to calculate the results of linear algebraic equations, respectively. All of developed equations and CFD procedure were then written as standalone computer program code. The developed CFD program can handle both symmetry and non-symmetry boundary conditions. Parameters used in CFD simulation are listed

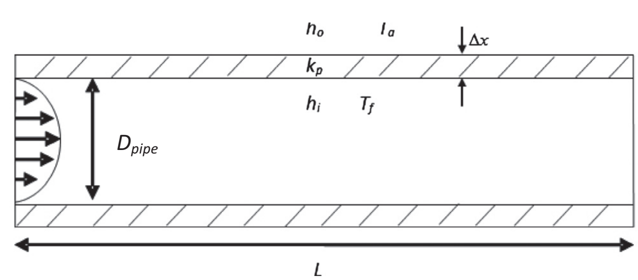

Fig. 1. The schematic drawing of a thick-wall straight pipe model.