Applied Thermal Engineering 117 (2017) 544–552

Contents lists available at ScienceDirect

Applied Thermal Engineering

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

Research Paper

# Two phase flow heat transfer analysis at different flow patterns in the wellbore

Yonghai Gao [a], Yanchun Cui [a], Boyue Xu [b], Baojiang Sun [a,*], Xinxin Zhao [a], Hao Li [a], Litao Chen [a]

[a] School of Petroleum Engineering, China University of Petroleum (East China), China
[b] Harold Vance Department of Petroleum Engineering, Texas A&M University, College Station, TX, United States

ARTICLE INFO

Article history:
Received 24 September 2016
Accepted 11 February 2017
Available online 16 February 2017

Keywords:
Wellbore
Heat transfer coefficient
Gas and liquid two phase flow
Flow pattern

ABSTRACT

Liquid gas two phase flow in the wellbore is a common phenomenon in oil and gas engineering. Flow patterns such as bubble flow, slug flow and annular-mist flow may significantly affect the heat transfer process. In this paper, heat transfer process in different flow patterns is studied by employing the thermal boundary theory. The Dittus-Boelter equation is modified by including the concept of gas volume fraction based on experimental data and new expressions for the heat transfer coefficient in different flow patterns has been proposed. It is found the heat transfer coefficient changes differently for different flow patterns. For bubble flow, the heat transfer coefficient increases with gas volume fraction. For slug flow, the heat transfer coefficient fluctuates largely and decreases as gas volume fraction increases. For annular-mist flow, the heat transfer coefficient rapidly decreases as gas volume fraction increase. In comparison with experimental data, the error range of the new expressions for heat transfer coefficient falls in ±20% which is good for field application.



## 1. Introduction

Multiphase flow heat transfer is important in oil and gas industry, nuclear industry and electric industry [1–3]. In drilling engineering and production engineering, especially during fluid flowing in the wellbore, there are many problems associated with multiphase flow heat transfer. Temperature distribution along the wellbore is an important issue for drilling safety control, especially in deepwater wells. If the temperature cannot be predicted and controlled properly, flow assurance problems will occur, such as, hydrate and wax deposition [4–8]. Fig. 1 illustrates the heat transfer process between wellbore fluid and surrounding environment in offshore wells. If the well is onshore, there is no sea water segment in Fig. 1.

During offshore drilling process, the heat transfer can be generally divided into two parts: the section above the mud line and the section below the mud line. Above the mud line, the heat transfer includes the heat convection between the drilling fluid in the drill pipe and the pipe inner wall, the drill pipe heat conduction, heat convection between drilling fluid in the annulus and the riser, the riser heat conduction and the heat transfer between the riser and the seawater. Below the mud line, the heat transfer includes the heat convection inside the drill pipe, the drilling pipe heat conduction, heat convection inside the annulus, the heat conduction of the casing, cementing and formation. During the producing process, the heat transfer has a little difference to the drilling, see Fig. 1 b. During drilling and producing processes, the total heat transfer resistance mainly includes: convection heat resistance inside the drill pipe $R_{ti}$, heat resistance of the drill pipe $R_t$, heat resistance of the annulus $R_a$, heat resistance of the casing $R_c$, heat resistance of the casing $R_{ce}$ [9–11]. If the flowing fluid inside the pipe is two phase flow, different flow patterns may occur, such as bubble flow, slug flow and annular-mist flow. Different flow pattern has different effects on the heat transfer mechanism and efficiency. According to the thermal boundary theory, with the fluid flowing up in the circular pipe, the thickness of thermal boundary layer increases. Therefore, the temperature field could be divided into two parts: first part of temperature difference in the thermal boundary, second part of same temperature. As there is no heat transfer in the second part, the heat resistance of forced convection is mainly distributed in the thermal boundary layer [12–14].

Fig. 2 shows that the heat transfer at different flow pattern is influenced by the heat resistance of the thermal boundary layer. In bubble flow, the heat resistance in thermal boundary layer is dominated by the liquid phase. Therefore, the heat resistance changes a little with gas void fraction increasing. In slug flow, due to the emerge of Taylor bubble, the heat resistance in thermal

* Corresponding author.
E-mail address: sunbj@upc.edu.cn (B. Sun).

http://dx.doi.org/10.1016/j.applthermaleng.2017.02.058
1359-4311/

**Nomenclature**

| | |
|---|---|
| $\Delta T_h$ | heated liquid temperature difference between inlet and outlet, °C |
| $\Delta T_c$ | cooled liquid temperature difference between inlet and outlet, °C |
| $\Delta P$ | heated liquid pressure differential between inlet and outlet, kPa |
| $Q_h$ | heated liquid volume flux, $m^3/h$ |
| $Q_c$ | cooled liquid volume flux, $m^3/h$ |
| $Q_g$ | gas phase flow flux, $m^3/h$ |
| $T_{hc}$ | heated liquid characteristic temperature, °C |
| $\rho_h$ | heated liquid density, $kg/m^3$ |
| $c_{ph}$ | heat capacity of heated liquid, kJ/(kg·K) |
| $\lambda_h$ | heated liquid thermal conductivity, W/(m·K) |
| $\eta_f$ | heated fluid viscosity at characteristic temperature, kg/(m·s) |
| $\eta_w$ | heated fluid viscosity at surface temperature, kg/(m·s) |
| $Pr$ | liquid Prantle number |
| $t$ | Temperature, °C |
| $\rho$ | Density, $kg/m^3$ |
| $c_p$ | heat capacity, kJ/(kg·K) |
| $\lambda$ | thermal conductivity, W/(m·K) |
| $\eta$ | Viscosity, kg/(m·s) |
| $\Phi$ | heat absorption of cooling liquid, W |
| $A$ | surface area of inner pipe, $m^2$ |
| $\Delta T_m$ | temperature difference of force convection heat transfer, °C |
| $c_{pc}$ | heat capacity of cooling liquid, kJ/(kg·K) |
| $m_c$ | mass flux of cooling liquid, kg/s |
| $h$ | heat transfer coefficient of forced convection, $W/(m^2 \cdot K)$ |
| $Re$ | liquid Reynolds number |
| $Nu$ | Nusselt number |
| $l_c$ | characteristic length, m |
| $C_t$ | temperature revised coefficient |
| $C$ | constant coefficient |
| $C_v$ | constant coefficient |
| $x$ | power of void fraction |
| $E_g$ | void fraction |

boundary layer consists of both liquid and gas phase, not only liquid phase. Thus, the heat resistance will increase with gas void fraction increasing, since the heat resistance of gas phase is larger than that of liquid phase. In annular-mist flow, the heat resistance in thermal boundary layer is composed of gas phase and liquid phase, which flows as liquid film along with the inner wall of pipe. When the gas void fraction increases, the heat resistance will increase, resulting in the decrease of heat transfer.

The research of liquid and gas two phase flow heat transfer focused on three areas. The first one was proposed by Rezkallah and Sims [15]. His liquid accelerating model assumed that the gas phase only accelerated the liquid phase rate and had no effect on heat transfer. So the two phase heat transfer was treated as single phase heat transfer and dominated by the average liquid rate rather than the two phase apparent flow rate. The second one was the pressure drop model proposed by Chu et al. [16] and Vijay et al. [17]. The pressure drop model assumed that the gas phase only had the effect on the liquid phase flow rate. The two phase pressure drop was obtained by the combination of single liquid and gas phase pressure drops. The single phase pressure drop was calculated by knowing apparent flow rate and fanning friction factor. The third one was compositional model which was presented by Knott et al. [18], Martin and Sims [19] and Shah [20]. In the compositional model, it was suggested that the heat transfer process was dominated by the effective flow rate of the mixtures, which was composed by the apparent flow rate of each phases. So the mathematical expression had the term of two phase Reynolds number.

In the past few decades, many experts had studied the gas and liquid two phase flow heat transfer and presented several heat transfer mathematical expressions to predict the wellbore temperature. Most of these mathematical expressions were built based on the given flow pattern. Kim et al. [21,22] concluded twenty expressions for heat transfer coefficient and analyzed the applicable conditions. Considering different liquid composition, flow rate, tube diameter and direction, Kim et al. [23,24] introduced the concept of effective wetted perimeter and presented the heat transfer coefficient expressions for horizontal and vertical two phase pipe flow. Based on this, Mahesh et al. [25] corrected the Nusselt number. However, most models did not consider the influence of thermal boundary and they worked with the specific flow pattern on the heat transfer process. For example, Kim et al. [22] concluded that for water–air flow within vertical pipes, Vijay et al. [17] correlation had a good performance that it takes 21 data points of total 25 within ±30% criterion in bubble flow. However, in slug and annular-mist flow, it did not satisfy the ±30% criterion very well, taking only 2/25 and 0/18. This paper analyzed heat transfer mechanism at different flow pattern and combined with thermal boundary theory and experimental simulation. New mathematical expressions for two phase flow heat transfer coefficient with cooling environment are presented.

## 2. Experimental setup and procedure

### 2.1. Introduction of the equipment

The experiment setup is shown in Fig. 3.

The experimental pipe flow consists of inner pipe and outer pipe, corresponding to the red and grey section in Fig. 3. The inner pipe simulates the casing pipe during drilling or oil tube during production process. Flowing fluid is gas and liquid two phase flow. It could simulate drilling fluid flowing after gas leak or two phase fluid flowing during production. The annulus, which is between inner pipe and outer pipe, simulates the cooling environment (formation or seawater). The drilling fluid or produced fluid (water and air) flow in the inner pipe, while cooling fluid flow in the annular. The experiment is able to simulate the heat transfer between the two phase flow in tube/annulus and the environment by adjusting annulus fluid temperature and inner pipe fluid temperature. The heat transfer is calculated by measuring inlet and outlet temperatures of inner pipe fluid and annulus fluid. The outer pipe is coated with insulated material and it is reasonable to assume there is no heat transfer outside the annulus.

The inner diameter (ID) and thickness of the inner and outer pipe is $\phi 60$ mm $\times$ 5 mm and $\phi 100$ mm $\times$ 5 mm, and their length is both 12000 mm. The material of inner and outer pipe is ordinary carbon steel, and their surface roughness is approximately 0.19 mm.

The inner pipe liquid flow rate is adjustable up to 10 $m^3/h$ and the maximum gas injection rate is 200 $m^3/h$. The bubble flow, slug flow and annular-mist flow were able to be observed within the ranges. The measuring experimental parameters include: liquid flow rate by electromagnetic flow meter; gas flow rate by mass