HEAT TRANSFER PERFORMANCE OF THERMAL-WASHING PROCESS FOR CRUDE OIL PIPELINE

by

Haiqian ZHAO, Zhenyu LU, Xiaoyan LIU*, Hui JIANG, Yang LIU, Lijun LIU*, and Fanbin MENG

School of Civil and Architectural Engineering, Northeast Petroleum University, Daqing, China

Original scientific paper
https://doi.org/10.2298/TSCI170918060Z

Study on the thermal wash multiphase flow and heat transfer characteristics have important significance to reduce energy consumption in wax removal. The thermal washing process of multiphase flow melting characteristics was simulated based on VOF model and RNG k-ε turbulence model. The melting temperature and pressure curves of wax were obtained under different initial conditions. The results show that the distribution of temperature in the pipe is uneven due to the action of gravity. Compared with the water temperature, the flow velocity has a greater influence on convection type. When the flow velocity is 0.5 m/s, the flow pattern of liquid wax is mainly suspended flow. When the flow velocity is 2 m/s, the wall flow is observed. The greater speed is or the higher temperature is, the faster melting rate is. The temperature increment varies with positions of the pipeline. A suitable condition for heat clean of wax was determined, which could save the cost of natural gas.

Key words: *thermal washing wax, numerical simulation, natural gas consumption, flow characteristic*

Introduction

The crude oil produced by various oilfields in China belongs to paraffinic base crude oil, which has high wax content, high pour point and high viscosity. In the process of crude oil extraction and transportation, when the oil temperature is lower than the wax precipitation point, the wax in the oil begins to crystallize, precipitate and deposit around the wall or the surface of the pipeline [1]. The main way to clean wax in our country is to use hot water to clean pipes. The hot water is usually heated by natural gas combustion in the oilfields. Changing the hot wash parameters can affect natural gas consumption. In the thermal wash process, the wax on the pipe wall absorbs the heat transferred by hot water, and the phase transition is induced, the melted wax is taken away by hot water. So the study of the hot water dewaxing process of phase change heat transfer and thermal fluid-flow laws has important significance to guarantee the safety of oilfield production and reduce natural gas consumption, while in the thermal wash process, the initial temperature and the wax layer changes greatly, but late changes are small. So it is important to master the law of change at the beginning.

* Corresponding author, e-mail: liu_xydq@163.com or lljdqpi@126.com

Different technological parameters have great influence on wax melting, especially temperature. At present, most people used the method of experimental to study on thermal wash cycle time, cost and other problems. And in theory, some scholars have studied wellbore temperature distribution by numerical simulation methods. Cheng [2] took the hot water wash cell as the research unit, and established the heat flow coupling model of the thermal washing process. The temperature distribution along the well depth was obtained by using iterative solution and cyclic calculation. Zhang [3] established a mathematical model for temperature distribution in the wellbore of hot oil washing process under consideration of formation leakage. Fan [4] established the vertical wellbore flow and heat transfer model by using the fluid mechanics theory and heat transfer principle. The wellbore temperature field model was established according to the law of energy conservation. The model was used to analyze the wellbore heating process and simulated the temperature of the thermal bath at different positions of the whole well by software.

In the process of crude oil pipelines washed with hot water, the paraffin on the pipe wall absorbs thermal energy and melts. The problem of phase change heat transfer is mainly concerned. Pal *et al.* [5] used the enthalpy-porous medium model to simulate the melting process of single heating wax in long rectangle with second boundary conditions. Assis *et al.* [6] regarded paraffin as the object of study and simulated the 3-D wax melting process with VOF model of FLUENT software, when the upper part of the sphere is exposed, which taking into account the effect of volume expansion. Tan *et al.* [7] established a 1-D mathematical model of paraffin melting under no slip limited condition in FLUENT software. Previous research studies on paraffin phase change energy storage have been carried out in detail, and most of them have been studied by numerical simulation. However, the problem of solid-liquid interface or natural convection of paraffin phase change itself is studied separately by the method of fixed boundary. Yet the flow and fluid-solid coupling interface problem of melted paraffin with other phase media is not involved.

In the study of fluidization of two-phase, Khanpour *et al.* [8] studied the mechanism of sediment transport induced by rapid flow through the meshless SPH model. Liu *et al.* [9] used VOSET method to simulate the deformation of gelled crude oil particles during hydraulic suspension transportation. The length and length of the deformed particles are determined by means of Image Pro Plus image processing software, and the size of the deformation is measured by deformation. Zhu *et al.* [10] studied the damage of submarine pipelines with different leakage sizes, and simulated with VOF method to predict oil spill flow distribution process with finite volume numerical shear velocity. Based on previous studies, it can be found that the VOF model can be used to study the flow pattern of multiphase flow.

The melt flow mainly includes flow and heat transfer problems of water or slurry, flow and heat transfer problems of molten metal. On the flow and heat transfer of pulp, Hayat *et al.* [11] studied the melting and homogenization/inhomogeneity effects in the extensional flow of a cylinder. The boundary-layer theory was used to explain the mathematical problems of velocity, temperature and concentration. Jiang [12] established a eulerian-eulerian two-phase flow model based on particle dynamics. The particle dynamics model is used to simulate the collision between particles and the interaction between particles and liquid phase turbulence. The turbulence model is used to take full account of the interaction between the liquid and the solid particles, and consider the melting of the slurry nitrogen and the phase momentum and energy transfer caused by the phase transition. Long [13] used VOF method and considered the effects of flow on the ice particle concentration, ice slurry thermal parameters and heat transfer properties of ice slurry ice slurry phase. A numerical model for phase change heat transfer of ice slurry in tubular fin heat exchanges is established. On the problem of heat transfer in molten metal flow, Yang *et al.* [14] had defined the momentum conservation equations including the effect of melt flow in laser drilling. The boundary-layer theory and integral method are used to obtain the product decomposition of the melt flow and the decomposition of the temperature field. The vapor pressure at saturated temperature is solved by using Clapeyron equation. Shabgard *et al.* [15] proposed CFD to analyze the melting of solid particles. Ma and Zhang [16] studied the heat transfer characteristics of phase change material slurry passing through a heating circular tube under constant heat flux, and discussed the characteristics of the evolution of slurry temperature. Liu *et al.* [17] studied CFD simulation of turbulent flow of metal bubble in an oblique blade propeller. Research on the flow of melting was very deep, but there is no research on crude oil pipelines hot wax removing process of multiphase flow problems.

Previous studies on wellbore temperature field, paraffin phase change, flow and flow melting have been studied, but further research is needed, which is on the crude oil pipeline heat removing wax process in multiphase flow, flow rate and wax melting interface problems. This paper focuses on the process of axial interface change. The VOF method is used to track the interfacial change of two-phase flow in paraffin removal process. The flow pattern and melting rate of multiphase flow under different thermal washing medium temperature and flow velocity are studied by numerical simulation.

Computational method

Physical model

In this paper, some kind of paraffin is chosen as the research object, and the paraffin parameters [18] are shown in tab 1. In the process of thermal washing, the wax layer changes under the action of hot water and flows out of the pipe along with the water. It is selected that the length of the calculation domain of 6000 mm and the height of 600 mm. The physical model is shown in fig. 1. The computational domain consists of two parts: the red section is the mainstream area, the main phase is hot water; the blue part is solidification zone, and the main phase is paraffin.

Table 1. Physical property parameter table of paraffin wax

Density [kgm^{-3}]	Specific heat [Jkg^{-1}K^{-1}]	Thermal conductivity [Wm^{-1}K^{-1}]	Viscosity [Pa·s]	Solidification temperature, [K]	Melting temperature [K]	Latent heat [Jkg^{-1}]
760	2100	0.25	0.000331	312	324	170000

Due to different positions along the length direction, the temperature rise is different. So in order to facilitate the analysis , it is divided into 6 regions that the model according to the length of the tube. Each cell size is 100 × 600 mm. The corresponding feature points are selected in each region: A:(–2500, –260), B: (–1500, –260), C: (–500, –260), D: (500, –260), E: (1500, –260), F: (2500, –260), and M: (–2500, 260).

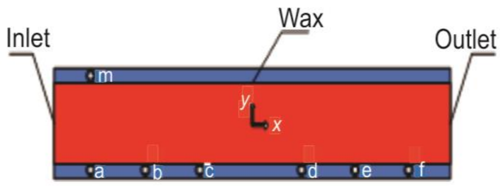

Figure 1. Skech map of physical model and characteristic points

The accuracy of the simulation depends on the quality of the mesh. In this paper, grid independent verification is performed for grids with 0.5, 1, 2, and 3 mesh spacing. Under the condition that the calculation accuracy is guaranteed and the amount of computation can be reduced, the mesh with 1 mesh spacing is finally simulated.

In this paper, the fully developed section of pipeline flow is simulated, and the flow of continuous phase water is calculated by RNG k-ε model. The inlet is set as velocity inlet, and the outlet is set as pressure outlet. The outlet pressure is set at atmospheric pressure and the wall is insulated.

Mathematical model

Hypothesis:

Paraffin wax is a single component substance, which is a constant substance. Due to the large number of turbulent Reynolds numbers in water flow, the natural convection of wax is very small, so the natural convection is neglected. The wall thickness is neglected. The viscosity of liquid paraffin and hot water is considered constant.

For two-phase flow simulation, the governing equations of VOF model are:

– *continuity equation*

$$\frac{\partial \rho}{\partial \tau}+\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}=0 \tag{1}$$

– *momentum equation*

$$\frac{\partial(\rho u)}{\partial t}+\frac{\partial(\rho uu)}{\partial x}+\frac{\partial(\rho uv)}{\partial y}=\frac{\partial}{\partial x}\left(\mu\frac{\partial u}{\partial x}\right)+\frac{\partial}{\partial y}\left(\mu\frac{\partial u}{\partial y}\right)-\frac{\partial p}{\partial x}+S_x \tag{2}$$

$$\frac{\partial(\rho v)}{\partial t}+\frac{\partial(\rho vu)}{\partial x}+\frac{\partial(\rho vv)}{\partial y}=\frac{\partial}{\partial x}\left(\mu\frac{\partial v}{\partial x}\right)+\frac{\partial}{\partial y}\left(\mu\frac{\partial v}{\partial y}\right)-\frac{\partial p}{\partial y}+S_y \tag{3}$$

– *energy equation*

$$\frac{D}{Dt}\left(\rho U+\frac{\rho V^2}{2}\right)=\rho Fv+\mathrm{div}(Pv)+\mathrm{div}(k\,\mathrm{grad}T) \tag{4}$$

– *the VOF volume fraction equation*

$$\frac{\partial \alpha}{\partial t}+v\nabla\alpha=0 \tag{5}$$

– *material property equation*

$$\rho=\alpha_1\rho_1+\alpha_2\rho_2 \tag{6}$$

$$\mu=\alpha_1\mu_1+\alpha_2\mu_2 \tag{7}$$

where $S = 0$, $S_y = -\rho g$, S_T is latent heat, ρ_1 – the density of water, ρ_2 – the density of wax, α_1 – the volume fraction of water within a unit grid, and α_2 – the volume fraction of wax within a unit grid.

For the wax layer:

– *the energy equation is*

$$\frac{\partial}{\partial t}(\rho T)+\frac{\partial(\rho uT)}{\partial x}+\frac{\partial(\rho vT)}{\partial y}=\frac{\partial}{\partial z}\left(\frac{k}{c_p}\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial z}\left(\frac{k}{c_p}\frac{\partial T}{\partial y}\right)+S_T \tag{8}$$

– *turbulent RNG k-ε equations*

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial(\rho k u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left(\alpha_k\mu_{\mathrm{eff}}\frac{\partial k}{\partial x_j}\right)+G_k+\rho\varepsilon \tag{9}$$

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\frac{\partial(\rho\varepsilon u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left(\alpha_\varepsilon\mu_{\text{eff}}\frac{\partial\varepsilon}{\partial x_j}\right)+\frac{C_{1r}^{'}\varepsilon}{k}G_k+C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \tag{10}$$

For the constants the following default values have been used:

$$\mu_{eff}=\mu+\mu_t,\quad \mu_t=\rho C_\mu\frac{k^2}{\varepsilon},\quad C_\mu=0.0845,\quad \alpha_k=\alpha_\varepsilon=1.39 \tag{11}$$

$$C_{1r}^{'}=C_{1\varepsilon}-\frac{\eta\left(1-\dfrac{\eta}{\eta_0}\right)}{1+\beta\eta^3},\quad C_{1\varepsilon}=1.42,\quad C_{2\varepsilon}=1.68 \tag{12}$$

$$\eta=\sqrt{2E_{ij}E_{ij}}\frac{k}{\varepsilon},\quad E_{ij}=\frac{1}{2}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right),\quad \eta_0=4.377,\quad \beta=0.012 \tag{13}$$

The commercial software FLUENT is applied to solve the governing equations with appropriate boundary conditions. The pressure and velocity fields are handled by the semi-implicit method for pressure linked equations algorithm. The standard scheme is used for pressure discretization, while the second order upwind scheme is used for momentum and energy equations.

Model validation

In order to validate the present numerical technique, numerical simulations are conducted for RT27_Rubitherm GmbH as the phase change material and compared against the predicted and measured results reported by Assis *et al.* [19]. A spherical shaped plastic EPCM of 40 mm in diameter is used. As the phase change material does not have a constant melting temperature, therefore a melting interval is used, with the solids and liquids temperatures set at 28 °C and 30 °C, respectively. In the simulation, the initial temperature of the computational model is 32 °C; however, the boundary condition is a specific temperature of 15 °C. The interface contours are predicated by using the present methods of enthalpy-porosity and volume of fluid simultaneously. These results are verified by the solidification process for paraffin by Assis *et al.* [19] to validate the numerical technique at different times such as 600 and 1500 seconds. It was concluded that the results qualitatively have a good agreement with results reported in. [19] Additionally, after 600 second the phase change material is 65% solidified and after 1500 second it is 89% solidified. According to the results presented by Assis after their phase change material was 95% solidified after 1500 second. [19] The agreement between the percent phase change material solidification further verifies the numerical technique in use.

Results and discussion

Effect of gravity on melting of paraffin wax

As shown in fig. 2, the heating rate of wax at the point A position is greater than that of point M. It is explained that due to the gravity, the melting of the region below the central axis of the tube is faster, and the temperature is larger than the central axis, resulting in uneven distribution of heat in the whole pipe section. 40 seconds ago, the temperature curve rises gently. At 40 seconds, there is only thermal conductivity at this point. Most of the heat is used

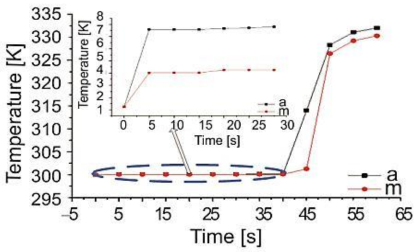

Figure 2. Curves of temperature change at point M and point A at a speed of 0.5 m/s and a temperature 333 K

for melting paraffin wax, with only a few passing through the point. The temperature changes between 40 and 50 seconds, because 40 seconds later, with the point of the outer wax melting and falling, there is convection and heat exchange at this point, so the temperature is abrupt.

Effect of different hot water parameters flow pattern of multiphase flow

Effect of flow velocity on flow pattern of multiphase flow

By analyzing the simulation results, the wax phase cloud images of different hot water parameters at different times are shown in fig. 3-5. In combination with fig. 3-5, when the water flow rate is 0.5 m/s, the wax layer slowly melts before 20 seconds, and the melted wax flows in the form of adhering wall. After 20 seconds, the flow pattern of the suspension began to appear. When the water flow rate is 1 m/s and 2 m/s, the flow pattern of multiphase flow is adherent flow after the beginning of melting. In order to consider the influence of convection conditions of different conditions more intuitively, the (1000, –240) points are selected as characteristic points, and the pressure variation curves at this point are shown in fig. 6.

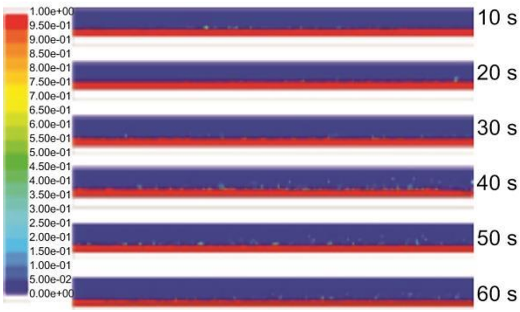

Figure 3. Local diagram of wax phase at 0.5 m/s and 333 K conditions

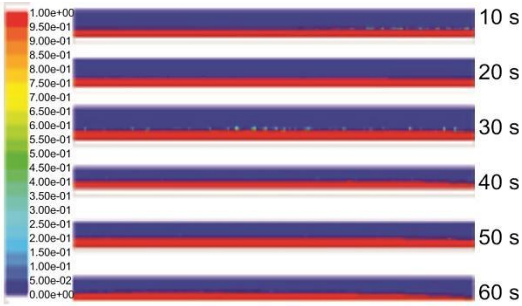

Figure 4. Local diagram of wax phase at 1 m/s and 333 K conditions

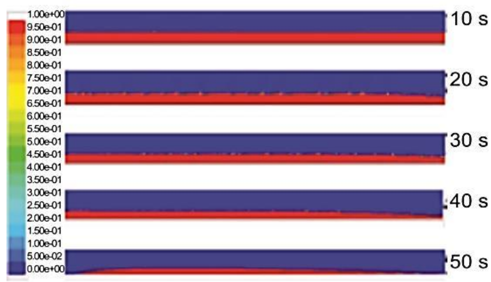

Figure 5. Local diagram of wax phase at 2 m/s and 333 K conditions

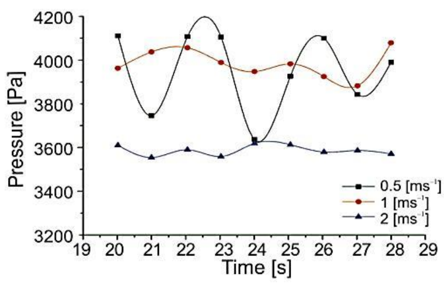

Figure 6. Pressure variation curve of point (1000, –240) at different speed conditions

In combination with fig. 3-6, the wax after melting does not diffuse into the main zone as the flow rate changes, and the liquid wax flows out of the pipe in the form of adherent or suspended flow. As can be seen from fig. 3, when the velocity is 0.5 m/s, the pressure fluctuation of this point is relatively large, indicating that there are many discontinuous and discontinuous liquid wax droplets at this point, and most of them drop in suspension form. When the speed is 1 m/s, the pressure value is relatively large, but the fluctuation is much smaller than that of 0.5 m/s, indicating that there is a uniform and continuous small droplets at this point, and according to the phase diagram, it is known that the flow pattern at this point is mainly adherent flow, and a few flow patterns are suspended flow. When the velocity is 2 m/s, the pressure value is not large and tends to be stable, indicating that the flow at the point is a wall flow, and the minor fluctuation of pressure is mainly due to turbulence. The shear force is proportional to the flow velocity. The greater the flow velocity is, the greater the shear force of the droplet is, so the flow velocity is 2 m/s and the flow state is in the adherent flow. When the flow rate is 0.5 m/s, the shear force is much lower than the buoyancy, so the resultant force of the shear force and the buoyancy can be approximated as buoyancy. But the length is not long enough, so the flow state is suspension flow.

Effect of temperature on flow pattern of multiphase flow

By analyzing the simulation results, the wax phase cloud images of different conditions and different times are obtained, as shown in figs. 8 and 9. In combination with figs. 5, 8, and 9, there is no fundamental influence of temperature convection type, *i. e*., the flow regime is a wall flow under three temperature conditions. Consistent with the section *Effect of flow velocity on flow pattern of multiphase flow* the point was selected as the characteristic point, and the pressure curve is shown in fig. 7. According to the phase diagram of fig. 7 and the corresponding temperature, the pressure fluctuates due to the flow of droplets sticking. When the temperature is 333 K, the pressure starts to fluctuate, but relative to the high temperature, the pressure value is smaller, because at this time the flow pattern is the wall flow, and the droplet distribution is uneven, but the droplet size gets smaller. When the temperature is 343 K, the pressure fluctuation is relatively stable and tends to decrease. The reason is that the size of the droplets is larger, and the droplet size and distribution tend to increase first and then decrease in that time range.

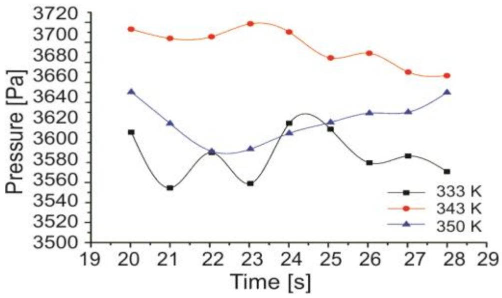

Figure 7. Pressure variation curve of point (1000, –240) at different temperature conditions

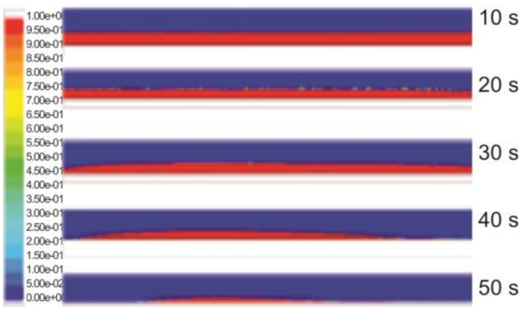

Figure 8. Wax phase diagram at 2 m/s and 343 K conditions

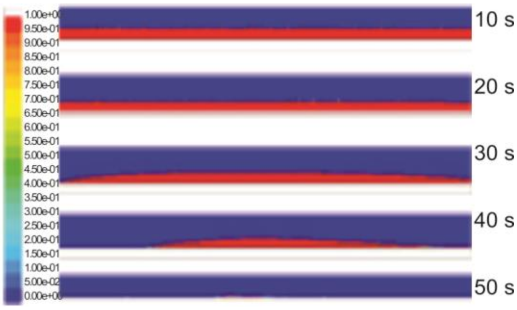

Figure 9. Wax phase diagram at 2 m/s and 350 K conditions

When the temperature is 350 K, the pressure decreases first and then continues to increase, because the droplet size decreases first and reaches near micron level, and then increases.

Effect of hot water parameters on wax deposition rate

Effect of velocity on wax melting rate

When the temperature is 333 K, the speed is 0.5 m/s, 1 m/s, and 2 m/s, respectively. The temperature variation curve is shown in fig. 10. According to fig. 10, the greater the velocity is, the faster the liquid fraction grows. The reason is that the velocity is directly proportional to the local heat transfer coefficient. The greater the local heat transfer coefficient is, the faster the melting rate is. However, the melting rate does not increase linearly with the flow rate. Between 1 m/s and 2 m/s, the melting rate increases with the flow velocity. Therefore, in practical engineering, the flow rate can be controlled between 1 m/s and 2 m/s, and the melting rate can be faster.

Effect of temperature on wax melting rate

When the speed is 2 m/s and the temperature is 333 K, 343 K, and 350 K, the temperature variation curve is shown in fig. 11.According to fig. 11, the higher the water temperature is, the faster the liquid fraction increases. Because the temperature difference is directly proportional to the local heat transfer coefficient, the greater the temperature difference, the greater the local heat transfer coefficient, the faster the melting rate, but not with the linear increase in water temperature. Between 343 K and 350 K, the melting rate increases with temperature. Therefore, during the actual wax removal process, the water temperature can be controlled between 70 and 80 °C, and the melting rate can be faster and the energy consumption can be reduced.

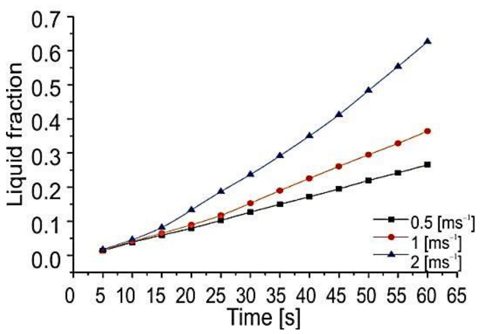

Figure 10. Liquid fraction curve diagram with different flow velocities

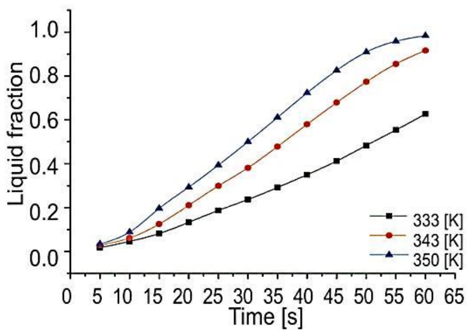

Figure 11. Liquid fraction curve diagram with different water temperature

Temperature variation at different positions

Select the speed of 2 m/s, the temperature of 333 K conditions, the temperature changes at each point of view, as shown in fig. 12. As can be seen from fig. 12, the different position of the temperature change along the different length of tendency, the other points of temperature rise rate from big to small: F > A > D > E > C > B. Due to the gravity, as shown in fig. 13, the pressure in the outlet portion is concentrated in the lower portion of the pipe, and the flow rate in the lower portion is increased. Therefore, the temperature at point F rises excessively. The entrance of such sudden pipe expansion phenomenon, the flow chart is shown in fig. 14, the entrance section of local convective heat transfer coefficient, temperature rise, and indirectly lead along the length direction, the local heat transfer coefficient is from big to small: F > A > D > E > C > B. Because of this condition, the whole period of time is the wall flow, so the wax after a point melting depends on the outer layer of the B point, which leads to the slow heating of the B point. Similarly, the point D also increases slowly due to the C point, and finally the temperature rise of each characteristic point tends to be stable.

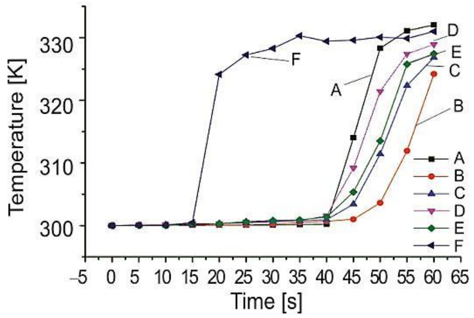

Figure 12. Temperature curve at different positions

Conclusions

Because of gravity, the region below the central axis of the tube melts faster and the temperature is higher than the central axis, resulting in an uneven spatial distribution of heat throughout the pipe section. Compared with temperature, the velocity convection type has a greater influence. When the flow rate is 0.5 m/s, the flow pattern of liquid wax is suspended. When the flow velocity was 2 m/s, the wall flow was observed. When the liquid wax is taken away by water, the local pressure is increased due to the viscous action. With the increase in velocity, the melting rate is faster, and with the increase of water temperature, the melting rate increases. In practical engineering, the velocity can be controlled at between 1 m/s and 2 m/s, and a faster rate of paraffin wax can be received. In the actual process, the temperature can be controlled between 70 ℃ and 80 ℃, fast melting rate and low energy consumption could be obtained. The different positions of the temperature change along the different length of tendency, the other points of temperature rise rate from big to small: F > A > D > E > C > B. The paper tentatively explores the influence of different parameters during the process of hot wash and wax removal on the effect of wax removal and lays the foundation for further research on more energy-efficient hot-wash parameters.

Figure 13. Pressure nephogram at exit section

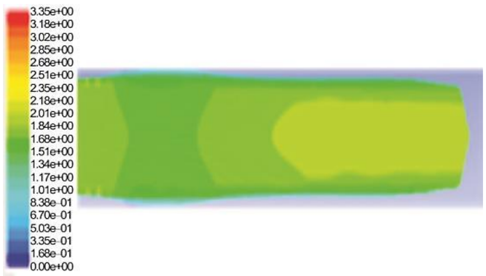

Figure 14. Entrance line streamline

Acknowledgment

This work was supported by National Natural Science Foundation of China (51534004) and Natural Science Foundation of Heilongjiang province (ZD2015011).

Nomenclature

c_p – air specific heat at constant pressure, [Jkg^{-1}K^{-1}]
F – volume force, [N]
K – kinetic energy of turbulence, [m^{2}s^{-2}]

P – pressure, [Pa]
T – temperature, [K]
u, *v* – flow velocity, [ms^{-1}]
W – width, [m]

Greek symbols

μ – fluid dynamic viscosity, [Pa·S]
λ – thermal conductivity, [$Wm^{-1}K^{-1}$]
ρ – fluid density, [kgm^{-3}]
τ – time, [s]
α – volume fraction, [%]
ε – dissipation rate of turbulent kinetic energy, [m^2s^{-3}]

Subscripts

i – microchannel
j – substrate
k – top cover
eff – inlet

References

[1] Li, C. X., *et al.*, Effect of Crude Oil Composition on Wax Deposition in Crude Oil Pipeline, *Journal of Chemical Engineering, 65* (2014), pp. 4571-4578
[2] Cheng, G. J., Study on Heat Flow Coupled Field and Temperature Monitoring System for Oil Well Thermal Washing, M. Sc. thesis, Harbin Institute of Technology, Harbin, China, 2010
[3] Zhang, Y., Numerical Calculation of Temperature Distribution in Wellbore of Oil Well during Thermal Washing Process, M. Sc. thesis, Northeast Petroleum University, Daqing, China, 2015
[4] Fan, J. W., Calculation and Application Research of Thermal Wash Temperature Field of Oil Well, M. Sc. thesis, Northeast Petroleum University, Daqing, China, 2010
[5] Pal, D., *et al.*, Melting in a Side Heated Tall Enclosure by a Uniformly Dissipating Heat Source, *International Journal of Heat and Mass Transfer, 44* (2001), 2, pp. 375-387
[6] Assis, E., *et al.*, Numerical and Experimental Study of Melting in a Spherical Shell, *International Journal of Heat and Mass Transfer, 50* (2007), 9-10, pp. 1970-1804
[7] Tan, F. L., *et al.*, Experimental and Computational Study of Constrained Melting of Phase Change Materials (PCM) Inside a Spherical Capsule, *Int. Journal of Heat and Mass Transfer, 52* (2009), 15-16, pp. 3464-3472
[8] Khanpour, M., *et al.*, Mesh-Free SPH Modeling of Sediment Scouring and Flushing, *Computers and Fluids,126* (2016), Apr., pp. 67-78
[9] Liu, X. Y., *et al.*, Numerical simulation of Particle Deformation in Gelled Crude Oil, *Journal of Engineering Thermophysics, 36* (2015), Mar., pp. 551-554
[10] Zhu, H. J., *et al.*, A CFD (Computational Fluid Dynamic) Simulation for Oil Leakage from Damaged Submarine Pipeline, *Energy, 64* (2014), Jan., pp. 887-899
[11] Hayat, T, *et al.*, Impact of Melting Phenomenon in the Falkner-Skan Wedge Flow of Second Grade Nanofluid: A Revised Model, *Journal of Molecular Liquids, 215* (2016), Mar., pp. 664-670
[12] Jiang, Y. Y., Preparation of Slurry Nitrogen and Its Characteristics of Flow and Phase Change Heat Transfer in Horizontal Tubes, M. Sc. thesis, Shanghai Jiao Tong University, Shanghai, China, 2010
[13] Long, Y. W., Phase Change Heat Transfer Characteristics of Ice Slurry in Tube Fin Heat Exchanger, M. Sc. thesis, Dalian University of Technology, Dalian, China, 2016
[14] Yang, Y. Q., *et al.*, Melt Flow and Heat Transfer in Laser Drilling, *International Journal of Thermal Sciences,107* (2016), Sept., pp. 141-152
[15] Shabgard, H., *et al.*, Heat Transfer Analysis of PCM Slurry Flow between Parallel Plates, *Int. Journal of Heat and Mass Transfer, 99* (2016), Aug., pp. 895-903
[16] Ma, Z. W., Zhang, P., Modeling the Heat Transfer Characteristics of Flow Melting of Phase Change Material Slurries in the Circular Tubes, *International Journal of Heat and Mass Transfer, 64* (2013), Sept., pp. 874-881
[17] Liu, H., *et al.*, Three-Dimensional CFD Simulation of Bubble–Melt Two-Phase Flow with air Injecting and Melt Stirring, *International Journal of Heat and Fluid Flow, 32* (2011), 5, pp. 1057-1067
[18] Ke, B. B., Numerical Simulation and Heat Transfer Enhancement of Paraffin Phase Change Heat Transfer Process Outside a Circular Tube, M. Sc. thesis, Jiangsu University, Jiangsu, China, 2016
[19] Assis, E., *et al.*, Numerical and Experimental Study of Solidification in a Spherical Shell, *Heat Transfer, 131* (2008), 024502

Paper submitted: September 18, 2017
Paper revised: December 21, 2017
Paper accepted: December 23, 2016