CFD simulations of convective heat transfer in actively heated tube bundle

To cite this article: Ola Karar et al 2018 IOP Conf. Ser.: Mater. Sci. Eng. 458 012065

View the article online for updates and enhancements.
CFD simulations of convective heat transfer in actively heated tube bundle

Ola Karar, Sampath Emani, Ramasamy Marappa Gounder * and Hilmi Mukhtar
Chemical Engineering Department, Universiti Teknologi PETRONAS, 32610, Bandar Seri Iskandar, Perak D.R., Malaysia

marappagounder@utp.edu.my

Abstract. Active heating is used as a flow assurance technique to prevent the huge temperature drop of crude oil while transporting from wells to facilities, which creates a friendly environment for hydrate formation and solid depositions. In the present study, convective heat transfer in tube bundle with three internal tubes was investigated through computational fluid dynamics simulation. A three dimensional model for existing bundle was developed. The model described the heat transfer occurring inside the bundle, aiming to heat up or at least maintain the production fluid temperature. The heat transfer from heating pipe to the whole bundle can be visualized through the temperature contours and the temperature gradient at the outlets. The effect of heating medium temperature and flow velocity are studied in the developed model.

Keywords - Active heating; tube bundle; computational fluid dynamics

Nomenclature

| Symbol | Meaning |
|--------|---------|
| A      | Area \( [m^2] \) |
| \( C_p \) | Specific heat \( [J/kg.K] \) |
| H      | Source term |
| g      | Gravitational force \( [m/s^2] \) |
| K      | Thermal conductivity \( [W/m.K] \) |
| M      | Mass flowrate \( [Kg/s] \) |
| n      | Number of tubes |
| Nu     | Nusselt number |
| P      | Pressure \( [Pa] \) |
| Pr     | Prandtl number |
| Q      | Rate of heat transfer \( [w] \) |
| Ra     | Rayleigh number |
| T      | Temperature \( [K] \) |
| v      | Velocity \( [m/s] \) |
| \( \rho \) | Density \( [kg/m^3] \) |
| \( \tau \) | Shear Stress \( [Pa] \) |
| \( \mu \) | Dynamic viscosity \( [N.s/m^2] \) |

Subscripts

| Symbol | Meaning |
|--------|---------|
| f      | Film |

ICPEAM2018  IOP Publishing  IOP Conf. Series: Materials Science and Engineering 458 (2018) 012065  doi:10.1088/1757-899X/458/1/012065

Content from this work may be used under the terms of the Creative Commons Attribution 3.0 licence. Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI. Published under licence by IOP Publishing Ltd
1. Introduction
The presence of hydrocarbons, water and other impurities, in crude oil eases the formation of hydrates as crude temperature falls below the hydrate formation temperature when transported from wellhead to facilities through flowlines and risers. Hydrates are solid-crystalline structured compounds formed when methane and other light hydrocarbons are trapped within water crystals under high pressure and low temperature. Once formed, hydrates continue to grow and aggregate and act as flow obstruction that causes production losses and environmental issues with high costs of maintenance. A study reported three incidents associated with hydrate plugs where the elbow was ruptured and resulted in loss of lives and over USD 7 million in capital cost [1].

Another flow related issue is wax deposition which restricts the crude oil flow and forms a three dimensional structure known as crude oil gel [2]. Wax precipitates from the oil when the temperature falls below wax appearance temperature (WAT) and transport with the production fluid or build up on the pipe surface.

Offshore harsh environment has increased the need to find ways to mitigate the issues of hydrate formation, wax and solid depositions which was later introduced as flow assurance by the oil conglomerate Petrobras in the 1990’s [3]. Flow assurance techniques guarantee safe, economic and reliable flow throughout the whole field life. The main concern is to maintain the temperature of the oil above hydrate formation temperature and WAT.

1.1. Flowline Thermal Management

1.1.1 Chemical Injection. Pour point depressants (PPDs) are added to waxy crude oil at an optimum concentration to lower its pour point, viscosity, yield strength and the wax deposition rates [4]. PPDs once injected in the pipeline, co-precipitate with the wax crystals and disturb its structure or hinder the wax particles to grow further. Low dosage hydrate inhibitors (LDHIs) are more cost-effective compared to conventional thermodynamic inhibitors [5]. Lower concentration of LDHI is needed compared to the conventional thermal inhibitors. Thus, it reduces the injection rates and the volume required. However, there are some challenges associated with using LDHI as it is non-recoverable [5].

1.1.2 Insulation. Insulation of flow lines and risers also reduces the heat loss to seawater; therefore, it keeps the crude oil warm and prevents hydrate formation. Using insulated tubing system will significantly decrease the volume of inhibitors injected into the system [6]. Materials such as polypropylene, polyethylene and polyurethane are used for flowline insulation depending on the required thermal conductivity [7]. Insulation techniques have improved starting from the conventional pipe in a pipe system filled with gas or vacuum in the annulus. However, the double-walled pipe is not economic for longer pipes.

1.1.3 Direct and indirect heating. In addition to the aforementioned methods, thermal hydrate management includes electrical heating and hot fluid circulation; in the first method, heat is directly generated through electrical cables where anodes are used to provide electrical communication with sea water. The other method is indirect heating through hot fluid circulation, where the heat is transferred through convection and radiation. Hot fluid circulation can be used with the conventional pipe in a pipe configuration, the heating medium flows in the annulus of the carrier pipe to maintain or heat up the production fluid temperature and prevent hydrate formation. Another configuration is using heat circulation in tube bundle which consists of multiple internal pipes enclosed together in one carrier pipe. The space between the internal pipes can be filled with air, nitrogen or water [8]. Most common heating fluids are water and glycol solution. Glycol-water solutions proved to have a better efficiency in heat transfer, lower corrosion and good microbiological control properties [9].

Throughout the past decades, tube bundles proved to be the most economical, efficient and practical flow assurance strategy for hydrate and wax mitigation through combining hot fluid circulation with
cost effective insulation. Moreover, the carrier pipe or outer jacket provides a good resistance to the seawater hydrostatic pressure [10].

Several studies have contributed in solving the problems of flow assurance in industry using indirect active heating method. However, heat transfer processes employed in some situations and the heat transfer coefficient predictions require improvement. Thus, the present study aims to investigate the heat transfer in 3 internal pipes tube bundle through computational fluid dynamics modelling. The effect of the heating medium temperature and flowrate on total heat gained by the production fluid was studied in the present work.

2. Convective heat transfer in tube bundles

Throughout the years, both internal fluid flow within an enclosure and external convection in a body surrounded by an infinite fluid medium grasped the interest of researchers in heat transfer area. Among the early theoretical models for natural convection was the correlation developed by Churchill and Churchill [11] for vertical plates, horizontal cylinders and spheres, which proposed an expression for Nusselt number (for both laminar and turbulent flow regimes) as in:

\[
Nu^{0.5} = Nu_0^{0.5} + \left[ \frac{Ra/300}{1 + \left( \frac{0.5}{Pr} \right)^{9/16}} \right]^{16/9} \]

which gives satisfactory results for all \( Ra > 10^{-4} \)

Another simplified correlation describing convective heat transfer in tube bundles was developed by Tillman [12] for in-line arrays of tubes

\[
Nu_f = 0.057 Ra_f^{0.5} \quad (2)
\]

and equation (3) for staggered arrays

\[
Nu_f = 0.067 Ra_f^{0.5} \quad (3)
\]

Churchill and Churchill correlation was then taken as a basis for an experimental and numerical study on tube bundle active heating by Liu [13]. Convective heat transfer between the three internal tubes enclosed in a carrier pipe filled with water was investigated and Nusselt number of a given tube surface was expressed in terms of Rayleigh number for all the tubes and Prandtl number from the expression.

\[
Nu_i = c \left( Ra_i^{1/3} \right)^{1/3} \left( \prod_{j=1}^{n} Ra_j \right)^{d} \quad j \neq i \quad (4)
\]

where

\[
[f(Pr)]^{1/3} = 0.8829 - 0.0013628 (Pr - 21.85) \quad (5)
\]

A study was conducted for Gulf of Mexico field to investigate the thermal interaction between the pipes in a bundle system and the effect on overall thermal performance and fluid arrival temperature [14]. The bundle had 3 insulated flow lines encased in steel, it was reported that using active heating, the entire bundle is performing as counter flow tube heat exchanger and computer model was developed to simulate multiphase flow in the bundle. The results have shown that doubling the flow rate of the heating medium increased the crude oil temperature by 6.8°C [14]. Thermal performance of tube bundle depends on bundle configuration, production fluid and heating medium temperature and characteristics, flow properties and the fluid flowing in the annulus of the tube.

Theoretical and experimental investigations on heat transfer analysis through tube bundles are available in literature for steady and transient states. Zabaras and Zhang [15] studied the steady state and transient behavior for tube bundle system using six different configurations. Overall heat transfer coefficients were determined and the cool down performances of the six configurations were compared
to the single pipe in pipe. However, the natural convection in nitrogen filling the bundle annulus was excluded from calculations.

Sallehud-Din [16] developed CFD model to predict heat transfer in 3 meter long tube bundle with 4 internal pipes enclosed in a sleeve pipe with spacers at both ends to hold the internal pipes in place. The model provided steady state calculations for the tube bundle in horizontal, inclined and vertical positions and it was reported that increasing the inclination angle will increase heat loss to surroundings and thus, reduce the efficiency of the tube bundle [16].

Danielson and Brown [17] developed steady state analytical model to predict the thermal performance of a 3-lines tube bundle and temperature threshold for hydrate mitigation. An OLGA simulation was performed and the data from the analytical model was used to evaluate the number of grids and overall heat transfer coefficient.

3. Flow circuit
The one-meter long bundle consists of two production pipes (204 mm OD) and one heating pipe (101.52 mm OD) enclosed in a carrier pipe (453.43 mm ID). A stainless steel tube (507.38 mm inside diameter) then surrounds the carrier pipe. Tubes are kept in position between two 25mm thick acrylic resin (Perspex) sheets. The sheets are tied together by tie rods external to the bundle.

Figure 1 presents the flow circuit, where water is continuously added to the internal tubes and the outer cavity. The storage tanks are filled with water which flows into respective tubes through pumps, temperature controllers are fitted to the tanks associated with the internal tubes to maintain the desired temperature and when exiting the tubes, water is fed back to the tanks.

4. Computational fluid dynamics study
CFD is used in simulating various flow and complex heat transfer problems. The governing equations for the model are solved together for each node in the developed mesh to investigate the system

![Figure 1. Tube bundle flow circuit.](image-url)
thermodynamic behaviour. ANSYS fluent was used to simulate the flow inside the tube bundle and predict the heat transfer.

4.1. Geometry and Meshing
A three-dimensional model of the prescribed tube bundle is developed. Steady state pressure based solver was used with K-epsilon turbulence model. Material flowing inside the internal tubes was set to be water. Figure 2 shows the developed geometry using design modeller.

Figure 2. Three dimensional model of the tube bundle

Tetrahedral mesh was generated for the tube bundle with a total elements number of 0.46 million where governing equations are solved at each node. Boundary conditions were set to be velocity inlet at the tubes inlet and outlet at tubes end was set as pressure outlet. Figure 3 shows the developed grid.

Figure 3. Developed mesh

4.2. Governing Equations
The flow inside the tube bundle is governed by Navier Stokes equation which represents the heart of fluid modeling, derived from conservation of mass and momentum. The non-linear differential equations are solved together for each node in the developed mesh. Conservation of mass is represented by continuity equation.

\[
\frac{d\rho}{dt} + \nabla \cdot (\rho \mathbf{v}) = 0 \tag{6}
\]

Conservation of momentum:

\[
\frac{d(\rho \mathbf{v})}{dt} + \nabla \cdot (\rho \mathbf{v} \mathbf{v}) = -\nabla p + \rho \mathbf{g} + \nabla \cdot \mathbf{t} \tag{7}
\]
Conservation of energy:

\[ \frac{d(\rho \ C_p T)}{dt} + \nabla \cdot (\rho \ C_p u T) = \nabla \cdot (k \Delta T) + H \]  

(8)

Total heat transferred within a tube is calculated from:

\[ Q = \dot{m} \ C_p \ (T_{inlet} - T_{outlet}) \]  

(9)

Steady state pressure based solver was used in the developed model, energy equation was enabled to predict the heat transfer rates and Standard k-epsilon viscous model was used for better convergence. Simulations were run for 1000 iterations. Fluid flowing inside the 4 tubes was set to water to later be validated with the experimental results.

5. Results

CFD simulations predict the heat transfer inside the tube bundle and the outlet temperature of all the streams were obtained. Convective heat transfer takes place where the hot water in heating pipe loses heat to the annulus fluid and further heat or at least maintain the production fluid temperature. Figure 4 shows the temperature contours for the annulus in between the tubes in 4 planes starting from inlets till the end of the bundle. Results were used to calculate the overall heat transfer rate inside the tubes to further investigate the effect of the heating medium temperature and flow velocity on the total heat gained by the production pipe.

5.1. Effect of heating medium temperature

Simulations were run with different inlet temperatures for the heating line (323 – 363 K) and the rest of the tubes were kept at constant temperature of 313 K. It can be observed that increasing the inlet temperature gives higher heat transfer rates and consequently, higher outlet temperature of the production fluid. However, the length of the tube bundle limits the time for heat transfer to occur fluid inside the tube, thus, the temperature gradients are low. Total heat gained by production pipe at different heating medium inlet temperature is shown in figure 5.
5.2. Effect of heating medium flow velocity
The study covered 5 cases of heating medium flow velocity (0.03, 0.04, 0.05, 0.06 and 0.07 m/s). Low flow velocities were simulated to provide enough time for heat transfer to occur as the bundle is one meter in length. Higher flow velocities gave higher temperature gradient and thus higher overall heat transfer rates. Figure 6 shows the graph for the heating medium flow velocities against the heat gained by production tube.

6. Conclusion
A three dimensional CFD model was developed to predict the heat transfer rates in a tube bundle with three internal pipes. The outlet temperatures of all the streams to evaluate the heat transfer rates inside the tubes were obtained. Cases with different inlet temperatures and flow velocities of the heating pipe were run and it was observed that the higher the heating medium inlet temperature and flow velocity the more heat gained by production pipe and thus, higher arrival temperature of production fluid.

Acknowledgments
Authors of the study are grateful to Universiti Teknologi PETRONAS for providing support and facilities through the research grant YUTP 0153AA-E07.

References
[1] Davalath J and Barker J 1995 Hydrate inhibition design for deepwater completions SPE Drilling & Completion 10 pp 115–121
[2] Ellison B, Gallagher C, Frostman L and Lorimer S 2000 The physical chemistry of wax, hydrates, and asphaltene Offshore Technology Conf. 11963 Texas, USA
[3] Camargo R, Gonçalves M, Montesanti J, Cardoso C and Minami K 2004 A perspective view of flow assurance in deepwater fields in Brazil Offshore Technology Conf. 16687 Texas, USA
[4] Thant M 2012 Experimental investigation of natural convection heat transfer in bundle pipeline PhD Thesis Imperial College
[5] Huo H, Wang R, Ni H, and Liu Y 2014 An Experimental Study on the Synergetic Effects of Kinetic and Thermodynamic Gas Hydrate Inhibitors Petroleum Science and Technology 32 pp 1940-47
[6] Rubel M and Broussard D 1994 Flowline insulation thermal requirements for deepwater subsea pipelines SPE Annual Technical Conf. and Exhibition Texas, USA pp 193-201
[7] Bai Y and Bai Q 2012. Subsea Engineering Handbook (Gulf Professional Publishing)
[8] Thant M, Sallehud-Din M, Hewitt G, Hale C and Quarini J 2011 Mitigating Flow Assurance Challenges in Deepwater Fields using Active Heating Methods SPE Middle East Oil and Gas Show and Conf. 140997 Bahrain
[9] Brown L 2002 Flow Assurance: A π Discipline Offshore Technology Conf. 14010 Texas, USA
[10] Goodlad M 2013 Bundle technology for the future SPE Offshore Europe Oil and Gas Conf. and Exhibition Aberdeen, UK
[11] Churchill S and Churchill R 1975 A comprehensive correlating equation for heat and component transfer by free convection AIChE Journal 21 pp 604-606
[12] D’Orazio A and Fontana L 2010 Experimental study of free convection from a pair of vertical arrays of horizontal cylinders at very low Rayleigh numbers International Journal of Heat and Mass Transfer 53 pp 3131-42
[13] Liu L, Choudhury T, Hu B, Duggan N, Richardson S and Hewitt G 2006 Experimental studies of heat transfer in a tube bundle model Proc. of the Institution of Mechanical Engineers Part E Journal of Process Mechanical Engineering 220 pp 151-159
[14] Brown T, Clapham J, Danielson T, Harris R and Erickson D 1996 Application of a transient heat transfer model for bundled multiphase pipelines SPE Annual Technical Conf. and Exhibition Denver, Colorado pp 377-393
[15] Zabaras G and Zhang J 1998 Bundle-Flowline Thermal Analysis SPE Journal 3 pp 363-372
[16] Sallehud-Din M 2012 Prediction of Natural Convection Heat Transfer in a Pipeline Bundle PhD Thesis Imperial College London
[17] Brown L, Clapham J, Belmear C, Harris R, Loudon A and Maxwell S 1999 Design of Britannia’s Subsea Heated Bundle for a 25 Year Service Life Offshore Technology Conf. Houston, Texas