Evaluation of Internal Electrical Heater for Pipe Temperature Control Using Finite Element Analysis (FEA) Model

Bo Xu*[,1], Marcus Heydrich[1], Stephen Edmondson[1], Alireza Jahangir[1]
[1]ShawCor Ltd., Toronto, ON, CANADA
*Corresponding author: 25 Bethridge Rd., Toronto, ON, CANADA, Email: bxu@shawcor.com

Abstract: In oil and gas industries, electrical heaters are often used to control the internal pipe temperature. Specifically, electrical heaters are used to simulate the high temperature conditions that pipe coating system are exposed to and are used to evaluate their performance. One important criterion for such test, is to maintain a uniform temperature range during the test. However, our observations indicated that a uniform thermal distribution around circumferential direction was hardly achieved during some tests. The objective of this study was to specifically resolve this non-uniform thermal distribution in circumferential direction, and to evaluate the heat transfer of one design of the internal electrical pipe heaters, by simulating both conduction and convection heat transfer around the heater and the steel pipe. Our model was based on Finite Element Method, using COMSOL multiphysics with CFD module. The simulation results revealed that convection air flow occurred at top half section of the pipe inner and highlighted the shortcomings of the existing heater design that a uniform thermal distribution in the pipe may not be achievable under current design.

Keywords: Computational Fluid Dynamics (CFD), Heater design, Convection Heat Transfer.

1. Introduction

As a common practice in oil and gas industries, pipeline technologies, including insulation coating systems have intensely been studied to understand their operational limits of prevent any potential economical and environment risks due to pipeline failure. For instance, during both product development and the final product evaluation, pipes are required to be heated to test its thermal operational limits. Electrical resistant heaters are often used to test the performance of pipe coating systems at elevated temperatures, when an oil heating method is not a viable option. This test method has reliably been used in various industrial applications, and in consumer appliances. Several electric heater manufactures provide electric heater elements as well as customized electric heaters for custom-need applications. Generally, the main challenge for using the electric heater to control the pipe temperature, is the pipe's limited inner space and issues related to wire connection. Even more challenging is when the electrical heaters are to be heated and submersed in water with a hydrostatic pressure up to 300 bars and incubation period of 30 days continuously. Another limitation is the total number of electric wires connected into the pipe.

Therefore, to reduce the risk of failure due to multiple parts and their exposure to high temperature, our design strategy consisted of developing a simpler heater with minimum required parts.. Figure 1 shows the electric resistant pipe heater manufactured by the electric heater contractor.

Figure 1. Electric resistant pipe heater prototype

While the existing heater design worked optimally well to heat 8" steel pipe with steel pipe wall thickness of 7/8", using the same design to heat a pipe with a bigger OD was proved to be problematic, as the heater was unable to maintain a uniform temperature profile along the pipe circumferential direction. It is important to note that as one of the pipe test condition, a uniform temperature distribution in the steel pipe is required. Accordingly, the current work was undertaken to evaluate heat transfer performances of the designed internal electrical pipe heater (as shown in Figure 1) by simulating both the conduction and nature convection heat transfer around the heater. The study was based on multiphysics Finite Element

Method simulations using COMSOL multiphysics with CFD module.

2. Governing Equations

In the current application, the electric heater is used to heat an insulated pipe surrounded by 4 °C water flowing at very low speed. Figure 2 shows a photo of a heater in the pipe. Upon running the test, it was realized that temperature distribution was uniform along the pipe axial direction, but not in the circumferential direction. To address this issue, a half circle 2-D model was used to simulate the heat transfer across the pipe cross-section normal to the axial direction. The 2-D model is effective because the length of the pipe is about 20 times bigger than the diameter and the heater element is the same along the axial direction and the symmetrical heater structure.

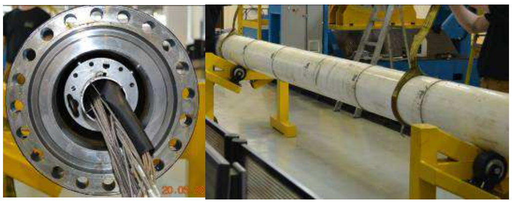

Figure 2. Electrical heater installed at inside of the pipe

Figure 3 shows 2-D geometric model of the heater used in the COMSOL model.

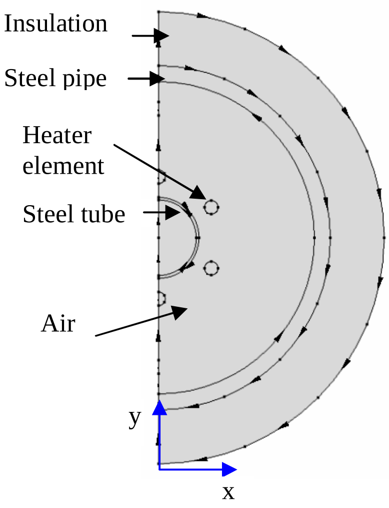

Figure 3. 2-D geometric model

In this model, the air flow and heat transfer is coupled through a temperature-dependent air density. Since the steady state heat transfer performance is of importance in this heater system, the governing equations for the steady state are provided by [1],

For air region,

$$\begin{cases} \dfrac{\partial(\rho u)}{\partial x}+\dfrac{\partial(\rho v)}{\partial y}=0 \\ \rho u\left(\dfrac{\partial u}{\partial x}+\dfrac{\partial u}{\partial y}\right)=-\dfrac{\partial p}{\partial x}+\mu\left(\dfrac{\partial^2 u}{\partial x^2}+\dfrac{\partial^2 u}{\partial y^2}\right) \\ \rho v\left(\dfrac{\partial v}{\partial x}+\dfrac{\partial v}{\partial y}\right)=-\dfrac{\partial p}{\partial y}+\mu\left(\dfrac{\partial^2 v}{\partial x^2}+\dfrac{\partial^2 v}{\partial y^2}\right)-\rho g \\ \rho c_P\left(u\dfrac{\partial T}{\partial x}+v\dfrac{\partial T}{\partial y}\right)=\dfrac{\partial}{\partial x}\left(k\dfrac{\partial T}{\partial x}\right)+\dfrac{\partial}{\partial y}\left(k\dfrac{\partial T}{\partial y}\right) \end{cases} \quad (1)$$

For the solid regions,

$$\rho c_p\left(\frac{\partial T}{\partial x}+\frac{\partial T}{\partial y}\right)=k\frac{\partial^2 T}{\partial x^2}+k\frac{\partial^2 T}{\partial y^2}+\dot{q} \quad (2)$$

The boundary conditions for the air flow are,

u = 0, v = 0 at the steel tube and the steel pipe,

$\dfrac{\partial u}{\partial x}=0$ at x = 0

The boundary conditions for the heat transfer are,

T = 210 °C at the heater element

$\dot{q}=\alpha(T_0-T)$ at the external surface of the insulation layer. Where α is heat transfer coefficient at the out surface of the insulation. T_0= 4 °C (in the real Simulated Service Vessel (SSV) test, water temperature is controlled to 4 °C).

$\dfrac{\partial T}{\partial x}=0$, at x = 0

To solve these equations, an extra fine triangular mesh was used as shown in Figure 4.

As shown in Figure 4, the complete mesh consisted of 5911 elements with 8831 as the number of degrees of freedom to be solved. In this COMSOL study, since the steady state temperature distribution in the pipe is of importance, a stationary study of a conjugate heat transfer model is used. At the solid region, a pure heat conduction mechanism exists. At the air flow region, a coupling between the air flow and heat transfer is also considered. The air flow is resulted from the thermal buoyancy effects due to the temperature gradient. A PARDISO solver with a nested dissection multithreaded pre-ordering algorithm is selected to solve the defined conjugated heat transfer study.

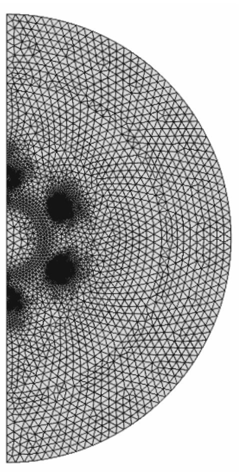

Figure 4. Triangle Mesh

Finally, the corresponding material properties used in the model (the properties of air use the built-in material library data) are shown in Table 1.

Table 1: Material properties used in the model

Material	ρ	c_p	κ	μ
Air	**	**	**	**
Steel	7850	475	44.5	NA
Insulation	690	1334	0.17	NA

*** data from COMSOL built-in material data*

3. Results and Discussions

Using the COMSOL model built, the air flow pattern induced by thermal buoyancy effects and the temperature distribution in the pipe system was computed, as shown in Figures 5 to 7.

Figure 5 shows the air flow induced by the thermal buoyancy effects due to heat released from electric heating elements. As shown in this Figure, air flow circles at the top 2/3 region of the pipe inner area. It starts from outer heating elements, flows upwards and around the center heater carriage tube, speeds up when it passes by the next heating element, turns downwards along the pipe inner surface, and then bounces back to upwards flow at the region slightly lower than the bottom heating element.

As known, there is a coupling effect between the air flow and the heat transfer when heating the pipe. At the equilibrium state, a flow pattern shown in Figure 5 exists in the system. This gives a temperature profile as shown in Figure 6. This Figure presents the 2-D temperature distribution across the pipe including the air at the inside of the pipe. As can be seen, at the inside of the pipe, the temperature at the heating elements is highest as given conditions, the region very close to heating elements is higher, and higher temperature area extends farther above the heating elements. The temperature is higher at the top region than that at the lower region.

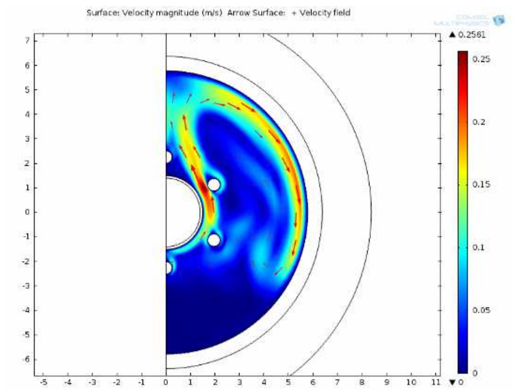

Figure 5. Velocity field of the induced air flow at the gap between heater element and pipe

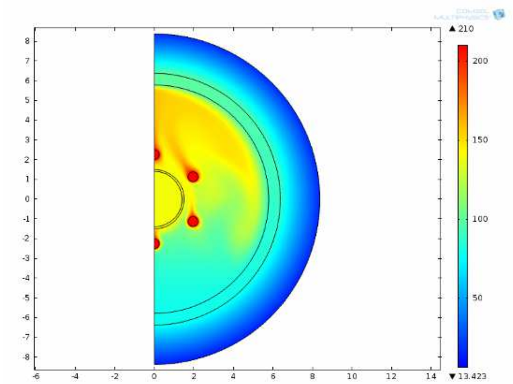

Figure 6. 2-D Temperature distribution in the pipe

Referring to Figure 5 and 6, it can be seen that the temperature distribution in the air region is directly affected by the air flow pattern. As the air flow passes by heating elements, the heat is carried and sent to the top region. This makes the temperature at the top region higher. The hot air is also sent slightly down when the air flows down along a narrow region close to the pipe inner surface. It makes the temperature at this narrow region is higher than that at its neighbor region. Such a temperature distribution in the air region results a non-uniform temperature distribution in the steel pipe region, where temperature is always higher at the tope region than that at the bottom region.

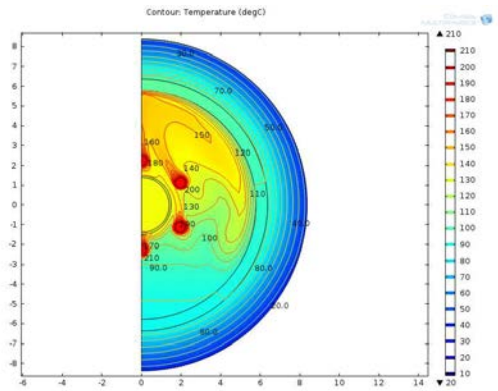

Figure 7. 2-D contour of temperature distribution in the pipe

To show the temperature difference in the system, Figure 7 presents a 2-D contour of the temperature distribution at the cross section (normal to the axial direction) of the pipe system. It is clearly shown that the temperature is non-uniform and aligns well with the flow pattern induced by thermal buoyancy effects. It can be seen that there is about 15 °C difference between the top surface and bottom surface of the steel pipe.

Due to high cost of carrying a heater test alone in the SSV system, a simple heater test rig as shown in Figure 8 was built to check the heater performance by attaching thermocouples at the steel pipe surface. It was observed that similar temperature difference between the top surface and the bottom surface occurred in the test setup when the heater elements are set to a constant temperature. Similar temperature difference were also seen during heater tests in the full scale pipe. This proved that the COMSOL model built provided a robust representation of the real design.

Figure 8. Electric heater test rig

7. Conclusions

A model using COMSOL Multiphysics together with CFD module has been presented to simulate the electric heater for heating a steel pipe. In this model, the air flow and heat transfer was properly coupled through a temperature depended air density. The model built performs well, giving results which agree well with experimental test results. It was concluded that the current heater design was not able to provide a uniform temperature profile at the steel pipe because of the nature convection heat transfer effects. This provided a solid foundation and reference for the future design of electric heater for heating pipe test.

8. References

1. Frank Kreith, Raj M ManglikAuthor, Mark S. Bohn, Principle of Heat Transfer, 74-75, 238-240, Cengage Learning, USA, (2011)

9. Acknowledgements

The authors thank Mark Redmond and Armen Markarian from Machine Development Center, ShawCor for their helps in the heater testing rig fabricate and set up.