Available online at www.sciencedirect.com

ScienceDirect

Energy Procedia 126 (201709) 18–25

Energy Procedia

www.elsevier.com/locate/procedia

72nd Conference of the Italian Thermal Machines Engineering Association, ATI2017, 6-8 September 2017, Lecce, Italy

Heat Exchange Numerical Modeling of a Submarine Pipeline for Crude Oil Transport

R. Lanzafame[a], S. Mauro[a]*, M. Messina[a], S. Brusca[b]

a Department of Civil Engineering and Architecture, University of Catania, Viale A. Doria 6, 95125, Catania, Italy
b Department of Engineering, University of Messina, Contrada Di Dio, 98166, Messina, Italy

Abstract

The present paper deals with a real issue of the Exxon-Mobil refinery in Augusta (Sicily). The crude oil, which is transported by oil tankers, is transferred through a submarine pipeline where it remains for a long time. In order to predict the transient temperature of the pipe, two numerical approaches were developed. The simplest one was a conductive model, based on the Finite Element Method, implemented by using the ANSYS Thermal FEM software for a first approximation solution. After having carried out an accurate grid resolution study and having evaluated the thermal error, a prediction of thermal profiles and heat fluxes was obtained. Thanks to the axisymmetrics of the physical problem, only a limited portion of the 3D pipe was modelled. The second approach was instead based on the use of a more accurate CFD Finite Volume Model, developed in ANSYS Fluent. In this case, in order to have reasonable calculation time and thanks to the aforementioned axisymetrics, the problem was carried out in 2D. Moreover, both grid and time step sensitivity was evaluated. Accurate buoyancy and turbulence models as well as viscosity and density temperature dependence models were used in order to obtain the most accurate physical modelling. The CFD model was developed basing on codes validated in the scientific literature. The comparison between FEM conductive and CFD results demonstrated the superior accuracy of the CFD, thanks to an accurate modelling of the internal convective motions.

Peer-review under responsibility of the scientific committee of the 72nd Conference of the Italian Thermal Machines Engineering Association

Keywords: Crude oil, pipeline, heat exchange, FEM Thermal, CFD

* Corresponding author. Tel.: +39-095-7382455; fax: +0-000-000-0000 .
E-mail address: mstefano@diim.unict.it

1876-6102 © 2017 The Authors. Published by Elsevier Ltd.
Peer-review under responsibility of the scientific committee of the 72nd Conference of the Italian Thermal Machines Engineering Association
10.1016/j.egypro.2017.08.048

1. Introduction

The problem of the crude oil cooling inside a pipeline is of the utmost importance for the oil industries due to the risk of reaching the pour point temperature. Specifically, the wax deposition or, even worse, the solidification of the crude, must be avoided in order to prevent the obstruction of the pipe. This would be an extremely complex and expensive problem to solve [1 - 4]. This issue is particularly evident in submarine pipelines where the presence of water drastically increase the heat exchange on the external pipeline surface. In the winter season, when the sea temperature is low and heavy oil are transported, the risk of obstruction is very high, above all if the oil remains in a standstill inside the pipe for a long time. Therefore, an accurate modelling of the pipe heat exchange will allow for a prediction of the transient thermal profile, thus helping the oil industries in the prevention of the obstruction risks. Specifically, the present paper deals with a real issue of the Exxon-Mobil refinery in Augusta (Sicily), where the oil, transported by tankers, is transferred to the ground through a long submarine pipeline. The refinery periodically needs to use heavy crude oils for economical reasons. These type of oil are often characterized by high pour point temperature (T_{pp} > 15 °C), thus the risk of pipe obstruction is very high in the winter season, when the sea temperature falls at nearly 15°C and the oil stands inside the pipe, waiting for the next tanker. A prediction of the time in which the oil reaches the pour point temperature is therefore essential for the refinery, in order to adequately program the unloading operations of crude oils.

Two different numerical approaches were used in this paper in order to obtain an accurate prediction of the time dependent thermal behavior of the pipe so as to provide an accurate estimation of the crude cooling time. The first approach was a simple conductive model, based on the Finite Element Method (FEM), applied to thermal simulations by using ANSYS FEM Thermal solver. The thermal model of the pipeline was generated on a reduced 3D domain with a length of 1 m, thanks to the axisymmetrics of geometry and physical conditions. The choice of a 3D domain, for a problem which might be studied in 2D, was due to the inherent tridimensional characteristic of the solver. The features of the materials and the boundary conditions, such as thermal conductivity, heat capacity, initial temperature and convective heat exchange coefficients were implemented using real data provided by the refinery. The inherent limitation with this modeling is that the fluid was considered like a solid, without internal motions. In order to evaluate the impact of the oil internal convective motions, which are due to the buoyancy effect, a more accurate CFD model was developed in ANSYS Fluent. The CFD model was generated by considering a 2D geometry so as to limit the computation time. Indeed, due to the transient characteristics of the problem and the small time-step needed, a full 3D CFD simulation would have required unreasonable computation time. This simplification was, however, again justified by the axisymmetrics of geometries and physical conditions.

Unfortunately, no experimental data were available for the real evaluation of the crude oil temperature inside the pipe, due to the impossibility to perform experimental measurements. However, the CFD model was developed following the scientific literature suggestions about previously validated CFD codes [5 - 7]. Specifically, the heat exchange between the oil and the walls can be easily and accurately calculated by the solver once the correct heat exchange coefficients and thickness of the materials are provided. Therefore, the only source of uncertainty in the CFD model might be the turbulence modeling related to the buoyant forces. In [8 - 10] was demonstrated that the RNG k-ε was the most suitable RANS model for the turbulence modeling of buoyant driven flows, thanks to improved terms which taken into account the generation of k and the production of ε due to buoyant forces. The RNG k-ε model was thus used in the present work. Furthermore, a grid independence study was carried out in order to ensure the minimization of the discretization errors. As the physical process to be simulated was inherently time dependent, a time step sensitivity study was made as well, thus finding the optimal time scale size for this particular physical problem.

2. Numerical Models

Both the numerical modeling strategies are presented in this section. As previously said, the FEM based conductive thermal model has an inherent physical limitation but can provide responses with very fast calculation time. The CFD model, instead, is certainly more physical accurate but, as will be seen hereinafter, it needs much longer calculation time. For this reason, it is worth to check out the validity of the FEM conductive model, even just to make a comparison with the CFD results so as to highlight the noticeable impact of the internal convective

motions on the crude oil cooling. Moreover, the possible effect of the radiation heat transfer can be easily and quickly evaluated by using the ANSYS Thermal model. In the present case study, however, the radiation can be neglected since its effect results two order of magnitude lower than that due to convection.

The main geometrical features of the pipe as well as the boundary conditions are the same for both the models and are summarized in Tab. 1. The pipe length is 1,200 m and it consists of an internal duct made by carbon steel, covered with a co-axial cylindrical layer made by reinforced concrete (Fig. 1). The pipe is placed inside the Augusta harbor at a mean depth of 14 m and is laid on the seabed, entirely surrounded by sea water. There are no relevant underwater currents so the water can be considered immovable. The crude oil to be simulated is a heavy oil, with API 37,4°. The oil physical characteristics were provided by the refinery and are presented in Tab. 1. The sea temperature varies between a maximum of 30 °C in summer and a minimum of 15 °C in winter. The simulations were thus carried out by imposing the minimum sea temperature (15 °C) as this was the most hazardous condition.

Table 1. Pipeline features, boundary conditions and crude oil properties (courtesy of Exxon-Mobil)

Pipeline Geometrical Features			**Crude Oil Properties**	
Underwater depth	14 [m]		API degree	37.4 [°]
Pipe length	1,200 [m]		Pour point	22,5 [°C]
Internal diameter	965.2 [mm]		Thermal conductivity	0.13 [W/mK]
External diameter	1,221.2 [mm]		Heat capacity	1,670 [J/kgK]
Total thickness	128 [mm]		Density at 15 °C	837.4 [kg/m^3]
Boundary conditions	**Internal carbon steel**	**External reinforced concrete**	Density at 20 °C	833.8 [kg/m^3]
Thickness	13 [mm]	115 [mm]	Density at 40 °C	819 [kg/m^3]
Thermal conductivity	52 [W/mK]	0.29 [W/mK]	Viscosity at 15 °C	0.01692 [Pa s]
Heat capacity	470 [J/kgK]	750 [J/kgK]	Viscosity at 20 °C	0.01384 [Pa s]
Density	7,870 [kg/m^3]	2,240 [kg/m^3]	Viscosity at 40 °C	0.006969 [Pa s]
			Initial temperature	38 [°C]

The convective heat transfer between the sea water and the external concrete surface was not directly solved. An average convective heat transfer coefficient was imposed on the external concrete surface, thus taking into account the external convection. This coefficient was estimated by calculating the Rayleigh number (Ra) and the Nusselt number (Nu) for the specific natural convection problem [11], thus obtaining:

$$h = \frac{k}{D} Nu = 452 \ \frac{W}{m^2 \ K} \tag{1}$$

In equation (1), k is the water thermal conductivity, D is the external pipe diameter and h is the average convective heat transfer coefficient.

Although the underwater currents might be neglected, the uncertainty related to the possible presence of forced convection, due to sea water flows, suggested the use of a conservative value for the convective coefficient. For this reason, a value h = 1,000 W/m^{2}K was chosen for both FEM conductive and CFD analyses. Anyhow, both the models allow for an easy change of this condition, in such a way that different values may be tested. A more precise evaluation of the convective heat transfer coefficient will be assessed in future works by making a complete CFD model which will directly model the external water convection around the pipeline.

2.1. FEM based conductive numerical model

The strategy to develop a FEM thermal model in ANSYS starts from the definition of the computational domain. As aforesaid, the axisymetrics of geometry and physical conditions allowed for the limitation of the study to a unit length of the pipeline (one meter). The geometry was simply made by three co-axial cylinders with the dimensions reported in Tab.1. Both the contact regions, between crude oil and carbon steel layer and that between carbon steel and concrete, were set as bonded perfect connections. The physical conditions for all the cylinders were properly set

so as shown in Tab. 1. Specifically, the appropriate values of thermal conductivities, heat capacities and densities were imposed to the solver. The intrinsic feature of the FEM conductive solver involved the fluid zone (Crude Oil) to be considered as a solid zone. For this reason the inner convective motions of the oil could not be taken into account.

The ANSYS FEM Thermal software directly solves the conductive and radiative heat transfer equations while the external convection is modeled by setting the appropriate convective coefficient. The equations are solved on the computational grid nodes. In order to ensure the minimization of the discretization errors, three mesh refinements were tested. The best compromise between accuracy and low computation time was already found with the coarsest mesh as the results did not significantly change with the refinements.

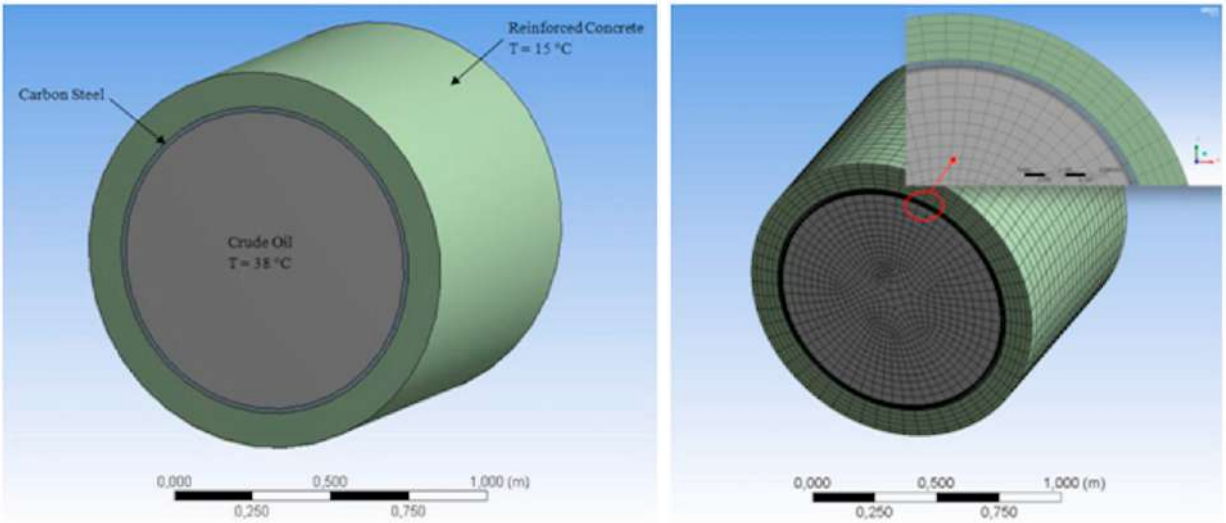

Fig. 1 Pipeline geometry and boundary conditions (left) and FEM conductive mesh with focus on the inflation prism layers (right)

The mesh, which was used for the present calculation, consisted of nearly 90,000 nodes and it was constructed by using an all quad element sweep method. In order to obtain a good resolution of the local gradients in the region of the carbon steel layer, local body sizing controls and prism layer inflations were used. The grid is shown in Fig. 1.

Due to the unsteadiness of the physical process, a transient thermal model was needed. In order to develop a transient simulation in ANSYS, an initial temperature distribution must be provided to the transient solver. This was done by coupling a steady state simulation, which calculated the temperature distribution at the initial time, to the transient model.

The time step size was automatically controlled by the solver which iteratively adapted the time step dimension, basing on the time scale of the problem. However, some tests were carried out in order to verify possible improvements with lower time step. The results showed that there were no improvements when lowering the time step, so the automatic control was used for the analysis.

The boundary condition for the convection was applied to the external concrete surface by using the aforementioned value of h.

An amount of 336 hours (14 days) were simulated with a computation time of nearly ten minutes.

2.2. CFD 2D numerical model

The computational fluid dynamic domain, in this case, was a simple 2D section of the pipeline and consisted of three concentric circumferences. The internal circle, which contained the crude oil, in this case was adequately treated as a fluid zone while the external layers were modeled as solid zones with the physical properties reported in Tab. 1.

In order to find the best compromise between accuracy and computation time, a grid independence study was carried out. Four mesh refinements were tested by controlling the size of the global surface mesh ($mesh_1$ = 0.1 m; $mesh_2$ = 0.05 m; $mesh_3$ = 0.01 m; $mesh_4$ = 0.005 m). The optimal compromise was found with the $mesh_3$ which had a maximum face size of 0.01 m and consisted of nearly 160,000 2D unstructured quadrangular elements. Specific refinements were done by placing inflation prism layers at all the interfaces (oil-steel; steel-concrete; concrete-sea water). Details of the mesh are reported in Fig. 2.

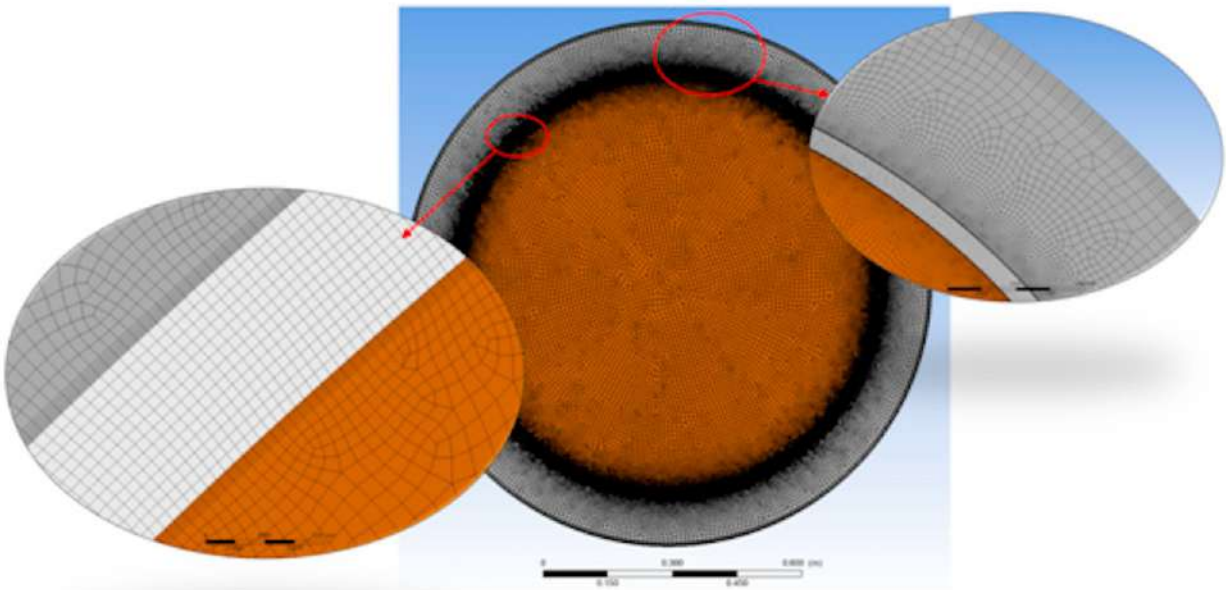

Fig. 2 CFD mesh with focus on refinement zones

The ANSYS Fluent Finite Volume software solves the Navier-Stokes equations and the continuity equation on the fluid zone. The heat transfer is resolved on both the solid and the fluid zones by enabling the energy equation. In order to model the buoyancy effects, which were responsible for the crude oil convective motions, the gravity was enabled and the density-temperature dependence was provided in Fluent using the data in Tab. 1. Moreover, the dynamic viscosity variation with temperature was implemented basing on the values reported in Tab. 1.

The unsteady RANS formulation was used for the turbulence modeling. Specifically, as previously said, the RNG k-ε model was implemented following the scientific literature suggestions [8 - 10]. This model inherently take into account the turbulence effects related to buoyant forces by using improved correlations in the production and destruction of k and ε in both the transport equations.

Since the physics of the problem was inherently time dependent, an adequate time step setting was of the utmost importance. The time step size must ensure that all the relevant physical time scales were captured by the model. For this reason, a time step sensitivity study was carried out by trying six different sizes. The results are showed in the charts below (Fig. 3). As can be seen in Fig. 3, the results are quite sensitive to the time step dimension, therefore the choice of an appropriate value is essential for the simulation accuracy. Since the difference between 2.5 s, 1 s and 0.1 s were very limited, a time step equal to 2.5 s was used in this work. In this way both the accuracy and the limitation of the calculation times were ensured.

A SIMPLE scheme was used for pressure-velocity coupling while a least squares cell based for gradient and a second order upwind algorithm were implemented for the spatial discretization of all the equations. Furthermore, a second order implicit time discretization algorithm was chosen.

The initial condition for the temperature distribution was provided by patching the initial oil temperature (38 °C) on the oil zone and performing an initial steady state simulation, subsequently switched to the transient analysis. An amount of 84 hours were simulated with a computation time of over five days.

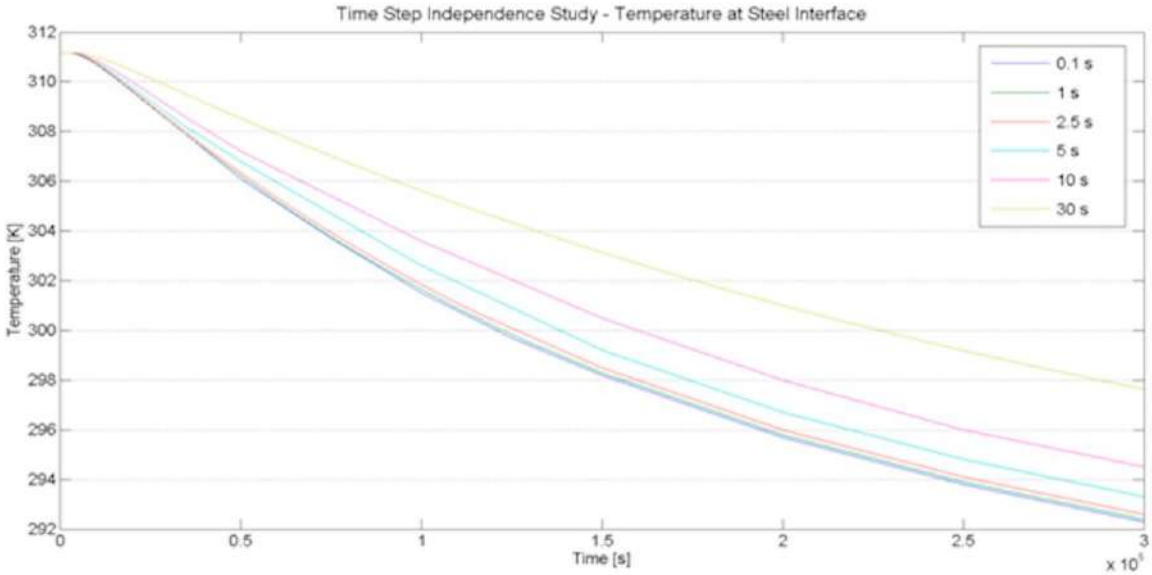

Fig. 3 Steel-Oil interface Temperature trend at different time step sizes

3. Results and comparisons

All the simulations were carried out on a HP z820 workstation with 128 Gb of RAM memory and two Intel Xeon E5-2695 processors with six cores for each and 24 available threads. Parallel computations were performed for both FEM conductive and CFD analyses. The strong differences in computation time are highlighted above. Only few minutes were needed for the FEM conductive calculations while the CFD model needed at least five days for the simulation of 84 hours of the thermal process. However, the superior physical accuracy of the CFD model is demonstrated by the results presented below.

Indeed, in the chart showed in Fig. 4, strong differences about the calculated time-dependent crude oil temperature trends are evident. Specifically, from the comparison of the minimum temperatures, the FEM conductive model would seem to predict faster oil cooling.

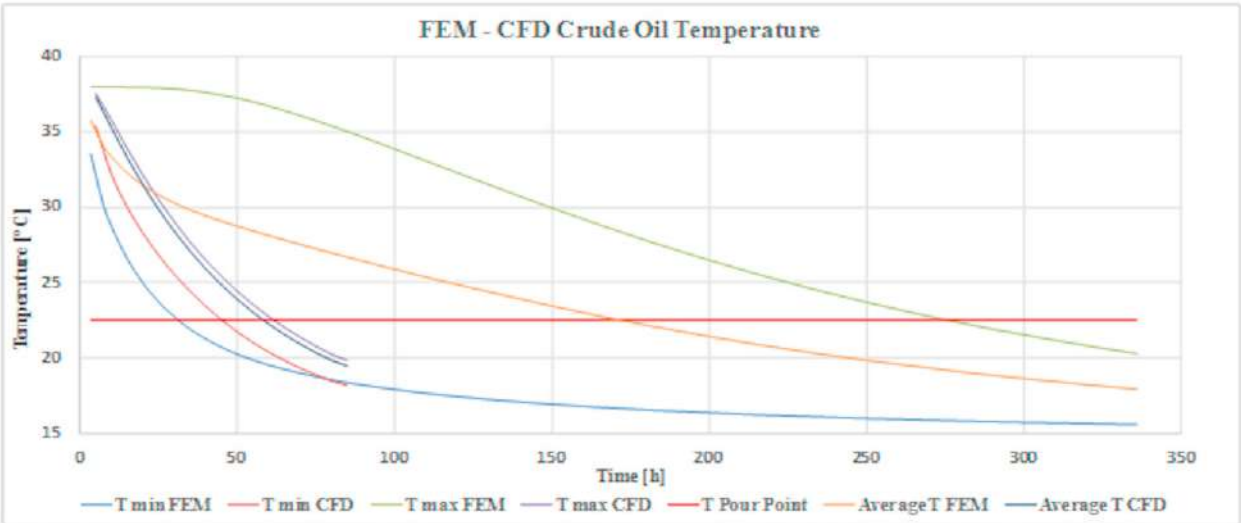

Fig. 4 Minimum, maximum and average temperature FEM conductive 50 - CFD trend comparison

However, Fig. 5, which presents the temperature distribution after 84 hours, shows a completely different physical behavior. The oil convective motions causes a mixing of the oil that cannot be taken into account by the FEM conductive analysis. This involves, in turn, a more uniform temperature distribution in the CFD simulation with the cooler oil concentrated at the bottom. Most of the crude oil is much cold than in FEM conductive results, thus indicating a global faster cooling due to internal convective motions.

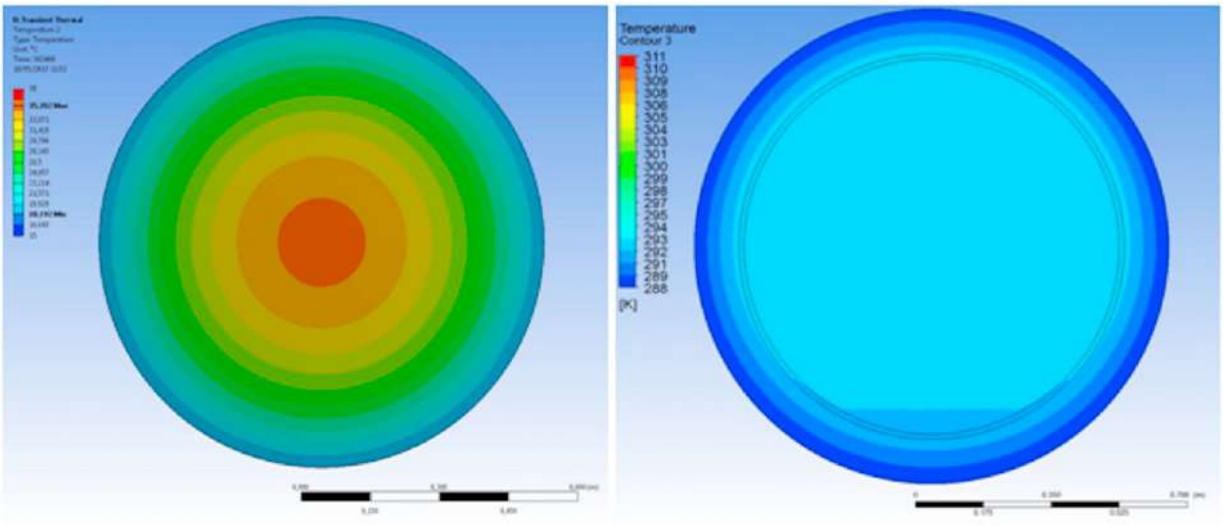

Fig. 5 FEM conductive (left) and CFD (right) temperature distribution at t = 84 h

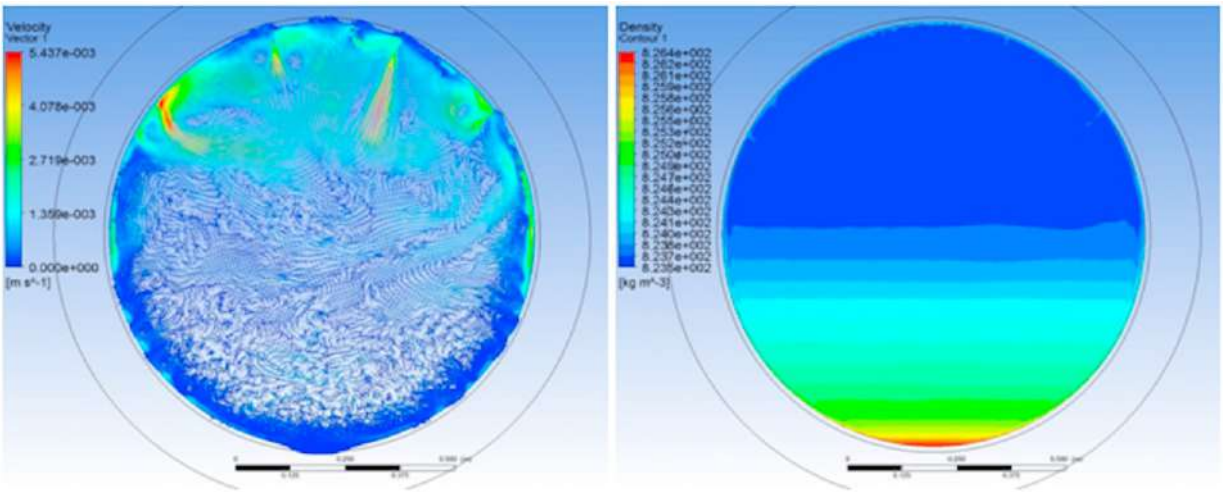

Fig. 6 CFD velocity vectors (left) and density (right) contours at t = 15 h

The inherent limitation of the FEM conductive modeling involves that the heat flux is modeled in a perfectly radial way with a concentric temperature distribution. For this reason, the oil near the steel interface is cooler than in the CFD. This is due to the fact that, as occurs in reality, the oil at interface is continuously replaced by warm oil while in FEM conductive the oil is immobile. Therefore, while the cold oil slides down along the steel wall, the warm oil is pushed upwards. This is further evident analyzing the average and maximum temperature in Fig. 4. The fast cooling, provided by the CFD model is highlighted and the average oil temperature reaches the pour point limit after nearly 60 hours while in the FEM conductive analysis the average temperature reaches the pour point after over 160 hours. Furthermore, the average temperature in CFD is very close to the maximum temperature and the overall gap between maximum and minimum CFD trend is very limited if compared to that obtained with the FEM

conductive model. This definitely shows the noticeable effects of the convective motions which generate a strong mixing of the fluid.

In Fig. 6 the velocity vectors and the density contours demonstrate the crucial effects of the buoyancy. At the top of the pipe, the buoyant forces generate little but significant velocities which result in the production of vortices as well (the estimated internal Grashoff number was $Gr \approx 8 \cdot 10^9$). Close to the lateral pipe surfaces, the oil presents a downward velocity which denotes the downward cold flow. Global intense scrambling phenomena are evident which are due to the turbulence caused by the buoyant forces. The chaotic flow behavior, showed in Fig. 6 (left), is just related to the unsteady chaotic nature of the buoyant turbulent motions. However, by time-averaging the velocity, upward mean motions, approximately located near the center of the pipe and downward mean motions, close to the lateral pipe surfaces, were detectable, therefore confirming the internal convective mixing of the oil. The cold and dense oil stagnates, instead, at the bottom of the pipe as evidenced by the density contours in Fig. 6.

4. Conclusions

In this paper two different models were implemented in order to simulate the crude oil cooling inside a submarine pipeline so as to provide an accurate estimation of the time in which the fluid reached the pour point temperature. The results demonstrated that the FEM conductive thermal model was not suitable for this particular physical modeling due to the fact that, the inherent limitation of the simple conductive model, did not allow for an accurate prediction of the oil buoyancy effects. The CFD model, instead, has proved to be much more accurate, with a precise prediction of the oil internal convective motions. The comparison between FEM conductive and CFD results showed that the buoyancy effects are of the utmost importance and must not be neglected in this kind of heat transfer problems. Indeed, the CFD model demonstrated that the convective effects inside the pipeline, generated a continuous and intense mixing of the crude oil which in turn caused a much uniform cooling, with the stratification of the cooler and dense fluid at the bottom of the pipe. This finally resulted in a global faster cooling prediction of the oil which reached the pour point temperature after nearly 60 hours against the over 160 hours obtained with the FEM conductive model.

Unfortunately, no experimental data were available for a direct validation of these models. However, the CFD model was based on codes which were widely validated in the scientific literature for more complicated cases.
The modeling strategy, presented in this paper, has proved to be a powerful tool and may be easily applied to the solution of similar physical problems.

References

[1] F. S. Ribeiro, P. S. Mendez, S. L. Braga "Obstruction of pipelines due to paraffin deposition during the flow of crude oils" Int. J. Heat Mass Transfer. Vol. 40, No 18, pp 4319-328, 1997

[2] Leiroz, A.T., Azevedo, L.F.A., "Studies on the mechanisms of wax deposition in pipelines" Offshore Technology Conference. Houston, 2005.

[3] A. Aiyejina, D. P. Chakrabarti, A. Pilgrim, M. K. S. Sastry "Wax formation in oil pipelines: A critical review" International Journal of Multiphase Flow 37 (2011) 671–694 doi:10.1016/j.ijmultiphaseflow.2011.02.007

[4] R. Martinez-Palou, M. de Lourdes Mosqueira, B. Zapata-Rendon, E. Mar-Juarez, C. Bernal-Huicochea, J. de la Cruz Clavel-Lopez, J. Aburto "Transportation of heavy and extra-heavy crude oil by pipeline: A review" Journal of Petroleum Science and Engineering 75 (2011) 274–282 doi:10.1016/j.petrol.2010.11.020

[5] W. Rukthong, "Development of computational fluid dynamics program for flow inside crude oil pipeline" M.S. thesis, Department of Chemical Technology, Faculty of Science, Chulalongkorn University, Bangkok, Thailand, 2014.

[6] W. Rukthong, W. Weerapakkaroon, U. Wongsiriwan, P. Piumsomboon, and B. Chalermsinsuwan, "Integration of computational fluid dynamics simulation and statistical factorial experimental design of thick-wall crude oil pipeline with heat loss" *Adv. Eng. Softw.*, vol. 86, pp. 49–54, 2015.

[7] W. Rukthong, P. Piumsomboom, W. Weerapakkaroon, B. Chalermsinsuwan "Computational Fluid Dynamics Simulation of a Crude Oil Transport Pipeline: Effect of Crude Oil Properties" ENGINEERING JOURNAL Volume 20 Issue 3 DOI:10.4186/ej.2016.20.3.145

[8] R. Kumar, A. Dewan "Assessment of Buoyancy-Corrected Turbulence Model for Thermal Plumes" Engineering Applications of Computational Fluid Mechanics, 7:2, 239-249, DOI: 10.1080/19942060.2013.11015467

[9] Shabbir, A., Taulbee, D. B., "Evaluation of Turbulence Models for Predicting Buoyant Flows" Heat Transfer J., vol. 112, 1990, pp. 945-951.

[10] Worthy, J., Sanderson, V., and Rubini, P., "A Comparison of Modified k-ε Turbulence Models for Buoyant Plumes" Cranfield University Library, Staff Publications, School of Engineering, 2001

[11] S. W. Churchill and H. H. S. Chu. "Correlating Equations for Laminar and Turbulent Free Convection from a Horizontal Cylinder" International Journal of Heat Mass Transfer 18 (1975), p. 1049.