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