ICHMT International Symposium on Advances in Computational Heat Transfer

May 28-June 1, 2017, Napoli, Italy

CHT-17-099

EFFECT OF GAS PIPE FLOW DIRECTION ON A PASSIVE SUBSEA COOLER EFFECTIVENESS: RESULTS OF 3D CONJUGATE HEAT TRANSFER SIMULATION

Nikolay Ivanov*, Vladimir Ris, Nikolay Tschur and Marina Zasimova
Peter the Great St.Petersburg Polytechnic University, St.Petersburg, Russia
*Correspondence author. Fax: +7 812 552 66 21 Email: ivanov_ng@spbstu.ru

ABSTRACT

For some applications, it is reasonable to design a passive heat exchanger based on buoyancy effects. A challenging example is a cooler aimed at subsea processing of natural gas if there is a requirement to reduce gas temperature. Several years ago, a concept involving gas compressors on the seabed in case of low reservoir pressure was suggested. Development of the subsea gas compression solutions requires, in particular, creating a cooling unit as reduced inlet temperature gives improved conditions for gas compression. To reduce servicing difficulties, the subsea cooler unit should be of a simple design with no moving parts. A promising way to meet the requirement of a robust subsea design is to organize a passive heat exchanger as an array of pipes that will use surrounding cold water as coolant. Heating of the external fluid by a bundle of parallel pipes will necessarily result in a buoyancy-induced draft flow, so that the bank inner-row tubes will be under conditions of mixed convection characterized by relatively low Reynolds numbers. Unlike forced convection conditions, if heat exchanger design uses natural convection as the main mechanism of cooling, heat transfer coefficient at the external surface of the pipe is one of the key uncertainties. Advanced CFD techniques could provide reliable data on external heat transfer for the low-Reynolds-number regimes typical for buoyancy-dominated coolers.

The current contribution deals with a model of a passive heat exchanger based on the buoyancy effects aimed at subsea gas processing of natural gas produced. The conjugate heat transfer model is used. Direct numerical simulation of external buoyancy-induced seawater flow through a staggered tube bundle at the Grashof number of 3×10^5 is coupled with the unsteady RANS modelling of internal natural gas flow at the Reynolds number of 8×10^5 and simulation of heat conduction through the steel pipe wall. A staggered tube bank composed of plain serpentine pipes of external diameter $D = 0.02$ m is considered. The pipe internal diameter $d = 0.014$ m is set. The computational domain includes two neighbouring pipes (two tube bank vertical rows) assuming the infinite number of pipes in the transverse direction with the periodicity boundary conditions at the vertical side boundaries. Numerical solutions were obtained with the commercial package ANSYS Fluent. The total grid size was 22.2 million cells. Time step of 0.02 seconds was used, the computed samples corresponding to statistically developed regimes were next to 300 seconds.

The goal of the study is to evaluate the effect of the pipe flow direction on the cooler effectiveness. The paper compares water flow and external heat transfer characteristics for two cross-flow cooler configurations with the downward and upward internal gas flow that correspond to the counter-flow and the parallel-flow heat exchanger schemes. It was found that though there is a pronounced difference in the local characteristics, in general the gas flow direction does not influence the cooler effectiveness: the total heat output values for both the schemes considered are almost the same. The cooler performance does not depend on the inlet and outlet collector location that gives some freedom in cooler design.

The work was supported in part by the Russian Foundation for Basic Research (grant No.15-08-02382).