Numerical simulation of conjugate heat transfer in a tube bank of a subsea cooler based on buoyancy effects

N Ivanov, V Ris, N Tschur, M Zasimova
Department of Aerodynamics, Peter the Great St. Petersburg Polytechnic University, Polytechnicheskaya str. 29, 195251, St. Petersburg, Russia
E-mail: ivanov_ng@spbstu.ru

Abstract. The contribution deals with a model of passive heat exchanger based on the buoyancy effects aimed at subsea processing of natural gas produced. 3D unsteady external buoyancy-induced seawater flow through a staggered tube bundle at $Gr = 3 \times 10^5$ was simulated using the full Navier-Stokes equations with no turbulence model in combination with the unsteady RANS modelling of internal natural gas flow at $Re = 8 \times 10^5$ and simulation of heat conduction through the steel pipe wall. Evolution of internal and external flow and temperature fields as well as variations in the heat output from row to row are predicted. It was found that the conjugate problem formulation resulted in dramatic changes in draft flow characteristics and external heat transfer coefficient values as compared with the simplified formulation based on the isothermal pipe wall assumption.

1. Introduction
For some applications, it seems reasonable to design a passive heat exchanger based on the buoyancy effects. A challenging example is a cooler aimed at subsea processing of natural gas produced if there is a requirement to reduce gas temperature.

The concept of subsea field development was suggested in the early 1970s, and since then subsea equipment application has advanced dramatically [1]. Several years ago, a concept involving gas compressors on the seabed in case of low reservoir pressure was suggested [2]. According to literature data, pioneering technology of subsea gas compression has been already installed successfully for the first time and started to operate since September 2015 [3]. Development of the subsea gas compression solutions requires, in particular, creating a cooling unit as reduced inlet temperature gives improved conditions for gas compression. As the subsea production and processing systems move into deep water with increase in servicing difficulties, the 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 [4]. 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 [5], 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 Computational Fluid Dynamics (CFD) techniques could provide reliable data on external heat transfer for the low-Reynolds-number regimes typical for buoyancy-dominated coolers.
The contribution by Ivanov et al. [6] is the first example known from literature of a CFD study of a tube bank representing a subsea cooler. Parametric computations of 2D unsteady buoyancy-induced flow through a staggered tube bank of 24 rows were performed in [6] to estimate the effects of the transverse and longitudinal pitches on the heat removal rate. The main conclusion was that even in case of 2D formulation used, for some configurations the external heat transfer computational data differ significantly from the forced convection measurement data available in literature. 3D turbulent flow through a network of pipes representing the subsea cooler tube bank was computed in [7] using a RANS turbulence model. The paper comprises a sensitivity analysis to investigate the effect of offsetting the pipes and lifting the cooler higher off the seabed. Small influence of cooler arrangement on the overall heat transfer was detected in the parameter range considered. 3D unsteady mixed convection through an inline tube bank of 12 rows was computed in [8]. The focus in the paper was on the effect of the superimposed forced side flow representing a sea current on external heat transfer. The computational results showed that a sea-current flow destroys the buoyancy-induced flow and therefore reduces the integral Nusselt number. Ivanov et al. [9] considered 3D buoyancy-induced flow through the same staggered tube bundle that was considered previously in [6] using 2D formulation. It was shown that the draft flow arising due to heating of the external fluid by the bundle of pipes produces strong ejection through side boundaries of the tube bank.

The current contribution deals with the same staggered tube that was considered previously [6, 9] using the assumption of a uniform temperature of the tube walls. In the current paper a more realistic conjugate heat transfer model is proposed: the external buoyancy-induced seawater flow through the tube bundle is computed together with the internal natural gas pipe flow.

2. Problem definition and computational aspects

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 illustrated partially in figure 1 (a), (b) includes two neighbouring pipes (two tube bank vertical rows \#1 and \#2) assuming the infinite number of pipes in the transverse \( y \)-direction with the periodicity boundary conditions at the vertical side boundaries. Each pipe contains 12 straight (horizontal) sections and 11 U-bends, so that the number of staggered horizontal rows in the tube bank is equal to 24. The longitudinal direction of the tube bank is parallel to the buoyancy force direction (\( z \)-axis in figure 1). The non-dimensional horizontal spacing \( a = 2.61 \) was set, \( a = S_1/D \), \( S_1 \) is the transverse pitch. The U-bend radius chosen provided the non-dimensional vertical spacing of \( b = 1.6 \), \( b = S_2/D \), \( S_2 \) is the longitudinal pitch. The height of the tube bank, i.e. the distance between the pipe inlet and outlet sections is equal to \( H = 0.8 \) m.

The straight section length value of \( L_{hor} = 32.5 \cdot D \) was assumed. The external boundaries of the computational domain in \( x \)- and \( z \)-directions were located at the distance of 50.0 \( D \) from the tube bank (not shown in figure 1). Preliminary computations resulted in a conclusion that this distance is enough to provide a domain-independent solution.

The physical properties were assumed to be constant. For water: \( \rho_f = 997.1 \) kg/m\(^3\), \( \mu_f = 8.9 \times 10^{-4} \) kg/m\( \cdot \)s, \( \lambda_f = 0.611 \) W/m\( \cdot \)K, \( C_{pf} = 4180.9 \) J/kg\( \cdot \)K, \( \beta_f = 2.057 \times 10^{-4} \) 1/K (the values correspond to 25\(^\circ\)C [10]). For steel pipe: \( \rho_s = 7840 \) kg/m\(^3\), \( \lambda_s = 12.9 \) W/m\( \cdot \)K, \( C_s = 482 \) J/kg\( \cdot \)K [11]. For natural gas at pressure of 40 bar and mean temperature of 35\(^\circ\)C: \( \rho_g = 35 \) kg/m\(^3\), \( \mu_g = 1.2 \times 10^{-5} \) kg/m\( \cdot \)s, \( \lambda_g = 0.04 \) W/m\( \cdot \)K, \( C_{pg} = 3000 \) J/kg\( \cdot \)K. The correspondent water Prandtl number is \( Pr_f = 6.09 \), and the gas Prandtl number is \( Pr_g = 0.9 \).

For the pipe flow, the uniform inlet velocity of \( V_{in} = 20 \) m/s was set that corresponds to the mass flow rate of 0.108 kg/s (per one pipe), the Reynolds number value is \( Re_g = 8.17 \times 10^5 \). The gas temperature value at the inlet was set as \( T_{in} = 50 \)\(^\circ\)C. For the water flow, the flow rate was not prescribed by the problem formulation being an output of the computational run; the uniform pressure was assumed at the external boundaries far from the cooler. The water temperature there was assumed as \( T_0 = 20 \)\(^\circ\)C. This temperature value corresponds to the conditions for a test (pilot) system in summer.
In addition to the above mentioned conjugate formulation, a non-conjugate problem has been solved assuming the isothermal boundary condition at the external pipe surfaces. The time- and area-averaged temperature value extracted from the conjugate problem solution was set at the walls, \( <T_w> = 34^\circ C \). The Grashof number based on \( D \) and on the reference temperature difference \( \Delta T_{ref} = <T_w> - T_0 \) is \( Gr = 2.8 \times 10^5 \). It is evident that on the average the free convection effects for the conjugate and non-conjugate convection could be characterised by the same value of \( Gr \).

External buoyancy-induced water flow was simulated on the basis of the unsteady formulation using the full Navier-Stokes equations with no turbulence model. The buoyancy effects were represented with the Boussinesq approximation. The internal natural gas flow was simulated using the unsteady Reynolds-averaged Navier-Stokes equations closed with the SST \( k-\omega \) turbulence model coupled with the wall function approach. Heat transfer through the steel pipe was simulated on the basis of 3D unsteady heat conduction equation.

The total grid size was 22.2 million cells. For the external water flow domain, the hybrid grid of about 19.4 million cells consisted of hexahedral mesh elements in most regions, namely in the near-wall boundary layers, everywhere between the straight sections of the pipe, and in the external part of the computational domain. Tetrahedral mesh elements were used in the inter-tube space near the bends. The grid was strongly clustered to the pipe surfaces: the height of the first near-wall cell from the external side was equal to 0.02 mm. From the internal side, the non-dimensional distance from the centre of the first near-wall cell to the wall, \( y' \), was equal to about 100.

Numerical solutions were obtained with the commercial package ANSYS Fluent 14.0. The second-order spatial discretization option (QUICK scheme) was chosen. The temporal discretization was performed with the three-layer second-order scheme, and the time step of 0.02 seconds was used for the computations. The samples taken for velocity, temperature and heat transfer coefficient time averaging corresponded to statistically developed regimes and were next to 300 seconds for both conjugate and non-conjugate cases. Note the heat transfer coefficient pulsations were up to 20% of its mean values with the main period of 1-3 seconds.
3. Discussion of the passive heat exchanger performance

Formation of the buoyancy-induced draft flow in the region near the tube bank is illustrated in figure 1 (c). The figure shows the instantaneous vertical velocity isosurfaces. Strong lateral flow at the edges of the tube bank resulting in 3D vortex structures formation behind the U-bends is visible in the plot. Side effects lead to significant variation in the flow rate of the upward draft flow with the height of the cooler. The vertical velocity values at the bottom of the tube bank compartment are very low, about 1 cm/s, while in the top part of the tube bank the local vertical velocity values are about 5 cm/s. The flow around lower straight sections of the tubes is stable, while there are pronounced 3D vortices above the upper sections. The mean draft velocity for the whole tube bank is equal to $V_{\text{draft}} = 0.034$ m/s that corresponds to the Reynolds number value of $Re_f = 760$ (based on $D$). Remarkably that as the ratio $Gr/Re_f^2 = 0.5$, the natural and forced convection effects are comparable in the water draft flow.

![Figure 2](image_url)

**Figure 2.** Instantaneous (a, c, e) velocity magnitude and (b, d, f) temperature distributions at three tube bank vertical cross-sections normal to the straight sections of the pipe.
Patterns of the instantaneous velocity magnitude and temperature fields corresponding to the statistically developed regimes are presented in figure 2. The data are given for three vertical sections oriented normally to the pipe straight sections. For the cross-section cutting the centre of a pipe, the disordered flow pattern is detected in the upper part of the tube bank compartment, starting from the straight sections #8, see figure 2 (e). For the cross-section at \( x = 0.31 \) m the flow is under strong influence of the side effects, and the velocity pattern looks like the flow past U-bends, see figure 2 (a). The side effects still influence the flow at the section at \( x = 0.17 \) m.

Transition from stable to nonstationary flow corresponds to changes in the temperature field: thus in the middle of the tube bundle (at lower \( x \)) the local thermal plume from the lower straight section remains stable up to section #8, see figure 2 (f). Starting from the section #9 the plume becomes unstable due to water mixing that results in significant changes in heat transfer.

![Figure 3](image)

**Figure 3.** Internal gas flow at three normal cross-sections of pipe straight section #7: (a) velocity magnitude values, m/s; (b) streamlines of the secondary flow; (c) tangent velocity vectors; (d) temperature distributions, data for steel wall are also shown. (I) – \( x = -0.3 \) m, (II) – \( x = 0.0 \) m, (III) – \( x = 0.3 \) m.

Internal gas flow is illustrated in figure 3 where flow patterns and temperature distributions are given for the straight section #7. Note that though the unsteady RANS formulation was used, the 3D flow patterns are steady state, and they are similar for both the pipes included into the computational domain. The pressure drop for the pipe with 11 U-bends was approximately equal to ten.

It is visible that at a cross section just behind the U-bend, at \( x = -0.3 \) m, the gas flow pattern is essentially 3D, and intensive secondary flow consists of multiple vortices, see row I in figure 3. Downstream the U-bend (at \( x = 0 \) and \( x = 0.3 \) m), the secondary flow is simplified and reduced. The secondary flow structure changes to a single-vortex, and the tangent velocity by the end of the straight section decreases almost by an order, see row III, figure 3 (c).

The temperature distributions given in figure 3 (d) also point to intensive secondary flows due to numerous U-bends. Remarkably that the temperature distribution in the steel pipe wall changes much.
in the circumferential direction, and this tendency remains for all three sections given in figure 3 (d). The main reason for it is a pronounced circumferential non-uniformity in the internal and external heat transfer coefficient distribution. In general, the temperature field’s structure visible in figure 3 (d) is a strong argument to use the conjugate convection formulation instead of a simplified isothermal wall approach.

Figure 4 gives the vertical velocity and temperature evolution plots that illustrate typical fluctuations in the tube bank compartment, the data for three monitoring points located in the mid-section \( x = 0 \) above straight sections #1, #7, and #12, at the distance of \( 1.1D \) from the wall surface. The data plotted in figure 4 correspond to the statistically developed regime. It is evident, that there is almost no pulsations above the straight section #1, and the amplitude of the pulsations in the vicinity of straight section #7 is very small, as these sections correspond to the stable flow region (see flow fields given in figures 1 (c) and 2). It is not the case at the highest section #12, where the amplitude of velocity fluctuations is almost 100% of the vertical velocity mean value, see figure 4 (a). The amplitude of the temperature fluctuations reaches about 30% of the mean temperature difference, \( \Delta T_{\text{ref}} \), see figure 4 (b). The fluctuations above section #12 are non-periodic, with continuous power spectrum, but there could be detected the main characteristic frequency of the pulsations approximately equal to \( f = 0.5 \, \text{s}^{-1} \).

![Figure 4](image_url). Time evolution of (a) vertical velocity and (b) temperature at three monitoring points located at \( x = 0 \) above the straight sections #1, #7, and #12 of the first vertical row.

The main disadvantage of the simplified non-conjugate formulation is that it does not reproduce the gas temperature decrease along the pipe and the correspondent pipe wall temperature reduction. On the contrary, the conjugate formulation simulates this process directly, as it is illustrated in figure 5. The values of the difference between the time- and area-averaged surface wall temperature of the current straight section and the outer water temperature far from the cooler, \( T_0 = 20^\circ C \), are plotted in the figure. This temperature difference represents the external wall temperature variation with the height of the cooler. It is visible that the temperature increase with the vertical coordinate is slightly non-linear. It is visible also that at the same \( n \) (at \( n > 7 \)) the temperature values for the second vertical row are slightly lower than for the first row, as the straight sections for row #2 are located higher than the straight sections for row #1 that results in better heat transfer conditions. Note that for the lower straight sections at \( n < 6 \) the characteristic temperature difference in case of conjugate convection is lower than in the case of the non-conjugate formulation, while at \( n > 6 \) the temperature difference for conjugate convection is higher.
Figure 5. Temperature difference between the time- and area-averaged external temperature of the straight section $n$ and far-field water temperature, $T_0$; dashed line – averaging over the entire wall surface.

Figure 6. Integral heat output from the straight sections of each vertical row: a comparison of conjugate heat transfer problem data with the non-conjugate problem results.

Figure 6 gives data on variation in the heat output values for both the cases computed. For the simplified non-conjugate case with the isothermal wall assumption, the heat output distribution versus the straight section number is non-monotonic: the minimum heat output value of about 300 W per straight section is registered at sections #4 and #5. The water flow in the vicinity of these sections is laminar, and the thermal resistance reaches maximum value due to direct interaction of the upward warm water flow from the lower section of the bundle. Heat output from the lower sections (#1 to #3) is relatively higher as the temperature difference between the pipe surface and the neighboring water is higher, and natural convection effects play more important role. For the top sections (at $n > 7$) the heat output values become higher again as the transition from laminar to turbulent flow occurs. However, the heat output does not grow much, and at $n = 12$ $Q = 470$ W.

If conjugate convection formulation is used, the heat output values grow monotonically with $n$. An increase in thermal resistance values corresponding to laminar flow near the lower straight sections is compensated by the temperature difference growth. At higher $n$-values, heat transfer process becomes more intensive as the water flow tends to be turbulent, and at the same time the characteristic temperature difference becomes higher (at $n = 12$ $\Delta T = 18^\circ C$, and the correspondent value of heat output is $Q = 630$ W).

Note that despite significant difference in the local heat output characteristics, the total heat output value per one vertical row, $Q_{tot}$ is almost the same for both the problem formulations considered (to compute $Q_{tot}$, the heat output from the U-bends, $Q_{bend}$, was also taken into account; remarkably, that the percentage of the $Q_{bend}$ in the $Q_{tot}$ value is 13.3% that corresponds to the percentage of the area of U-bends in the total pipe area). For the conjugate convection formulation $Q_{tot, conj} = 5.25$ kW, while for the non-conjugate formulation $Q_{tot, non-conj} = 5.05$ kW, the difference is less than 4%. However, it is not possible to estimate the mean wall temperature, $<T_w>$, before the conjugate convection problem is solved, and the conjugate formulation is the only way to get reasonable heat transfer data.

4. Conclusions
Coupled model of a passive heat exchanger based on the buoyancy effects aimed at subsea processing of natural gas produced is considered in the study. External buoyancy-induced seawater flow through
a staggered tube bundle at the Grashof number of $3 \times 10^5$ was simulated on the basis of the unsteady formulation using the full Navier-Stokes equations with no turbulence model in combination with the unsteady RANS modelling of internal natural gas flow at the Reynolds number of $8 \times 10^5$ and simulation of heat conduction through the steel pipe wall. The conjugate convection data are presented in comparison with the non-conjugate formulation results.

For both conjugate and non-conjugate cases, a statistically developed regime was achieved, and a sufficient sample for time-averaging was collected. Evolution of internal and external unsteady flow and temperature fields as well as variations in the heat output from row to row are predicted.

It has been established that conjugate formulation leads to high non-uniformity of the vertical draft velocity that varies from 1 cm/s at the lowest row to about 5 cm/s at the top row. Strong non-uniformity of draft velocity due to side effects results in external heat transfer coefficient variation with vertical coordinate. Integral heat output values processed from computational data pointed to significant difference in heat transfer coefficient variation with vertical coordinate obtained with two different formulations.

Despite significant difference in the local heat output characteristics, the total heat output value per one vertical row is almost the same for two problem formulations considered. However, as it is not possible to get an adequate value of the mean pipe wall temperature before the conjugate convection problem is solved, conjugate formulation is the only way to get reasonable heat transfer data.

Acknowledgment

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

References

[1] Bai Y and Bai Q 2012 *Subsea Engineering Handbook* (Gulf Professional Publishing)

[2] Fantoft R 2005 Subsea gas compression – challenges and solutions *Offshore Technology Conference, 2-5 May 2005, Houston, Texas* (OTC paper № 17399) pp 1-6.

[3] The first subsea gas compression plant in the world on line – a step change in subsea technology 2015 Statoil news release URL: [http://goo.gl/ujieG0](http://goo.gl/ujieG0) (retrieved April 19, 2016).

[4] Gyles B R, Haegland B, Dahl T B, Sanchis A, Grafsronningen S, Schueller R B and Jensen A 2011 Natural convection – subsea cooling; theory, simulations, experiments and design *Proc. of the ASME 2011 30th Int. Conf. on Ocean, Offshore and Arctic Engineering, (OMAE2011 Rotterdam, The Netherlands, June 19-24, 2011)* pp 11-20.

[5] Zukauskas A 1989 *High-performance single-phase heat exchangers* (New York: Hemisphere Publishing Corp)

[6] Ivanov N G, Kirillov A I, Ris V V and Smirnov E M 2010 Numerical modeling of buoyancy-induced fluid flow and heat transfer in a staggered tube bank * Proc. of the Int. Heat Transfer Conf., IHTC14 (Washington, DC, USA, 8-13 August 2010)* pp 545-551

[7] Leahy M, Jagannatha D, Chauvet C and Holbeck J 2012 CFD modelling of a subsea cooler for calculation of external heat transfer coefficient *Proc. of the 9th Int. Conf. on CFD in the Minerals and Process Industries (Melbourne, Australia, 10-12 December 2012)* pp 1-6.

[8] Ivanov N G, Ris V V, Smirnov E M and Tschur N A 2014 Numerical simulation of 3D flow effect on heat transfer from a tube bank of subsea cooler *Proc. of the 15th Int. Heat Transfer Conf., IHTC-15 (Kyoto, Japan, 10-15 August 2014)* pp 1-12

[9] Ivanov N, Ris V, Tschur N and Yurkina N 2015 Numerical simulation of buoyancy-induced flow through a staggered tube bank of subsea cooler *Advances in Heat Transfer: Proc. of the 7th Baltic Heat Transfer Conf. (Tallinn, Estonia, 24-26 August 2015)* pp 23-28

[10] Gebhart B, Jaluria Y, Mahajan R L and Sammakia B 1988 *Buoyancy-induced flows and transport* (New York: Hemisphere Publishing Corp)

[11] Rolled Alloys Inc 2009 *Data sheet ZERON® 100 (UNS S32760)*