Effect of the bounding walls configuration on draft water flow through staggered tube bank

M A Zasimova, N G Ivanov, A D Podmarkova, V V Ris, N A Tshur

Department of Fluid Dynamics, Combustion and Heat Transfer,
Peter the Great St.Petersburg Polytechnic University,
29 Polytechnicheskaya Street, St.Petersburg 195251, Russia

E-mail: ivanov_ng@spbstu.ru

Abstract. The paper presents the numerical study of 3D unsteady buoyancy-induced water flow through a staggered tube bank typical for subsea cooling units used at seabed gas compression modules. For the Grashof number from the range of $10^5$...$10^6$ parametric computations have been performed using direct numerical simulation of the draft water flow in the inter-tube space. For various arrangement of the bounding walls, the effect of the side flow on heat transfer is analyzed and discussed.

1. Introduction

Natural gas compression near the production unit is required after some period of operation when the pressure in the reservoir is reduced. A compressor unit keeps the ability to transport the gas produced through a pipeline to the sea shore. A novel concept to install a compressor on the seabed close to the subsea wells was suggested about fifteen years ago [1]. Since then, a subsea gas compression unit has been once installed successfully on the seafloor of the Norwegian Sea [2]. One of the bottlenecks of the technology is the design of the inlet/discharge cooler that, being a necessary element of a subsea gas compressor unit, must be passive with no moving parts to reduce servicing difficulties.

One of the ways to design a passive heat exchanger is to set it as an array of pipes that will use surrounding cold water as coolant under natural convection conditions [3]. The problem is that if a heat exchanger operates with free convection as the main mechanism of cooling, there is high level of uncertainty in the heat transfer coefficient at the external surface of the pipe due to lack of literature data. Application of Computational Fluid Dynamics (CFD) techniques is a promising way to get reliable data on external heat transfer for low speed regimes typical for buoyancy-dominated flows.

3D turbulent flows through a network of pipes representing a subsea tube bank were presented for the first time in [4] using a RANS turbulence model and in [5] for a scaled cooler using a no-model approach also adopted in the current study. An advanced conjugate heat transfer model was used later in [6]: the external buoyancy-induced water flow through a tube bank was computed together with the internal natural gas pipe flow and heat conduction through the massive steel pipe wall.

One of the key factors of the cooler performance is ejection through side boundaries of the tube bank that could be very strong (see [7] and references there). All our previous studies [7] assumed the infinite number of pipes in the transverse direction with the imposed periodicity boundary conditions. To take into account side effects from all directions, it is necessary to consider a cooler entirely, with a finite number of pipes. According to [6], a simplified nonconjugate formulation is acceptable for parametric studies, and the current contribution returns to the isothermal wall approximation.
2. Problem formulation

2.1. Geometry model
The model of a staggered tube bank under consideration consisted of six vertical rows attached to the inlet and outlet collectors. Each vertical row is a plain serpentine pipe of external diameter \( D = 0.02 \text{ m} \). The tube bank has three lower (odd) pipes and three upper (even) pipes (figure 1b), the numeration of each pipe, \( N \), is shown in figure 1a using roman numerals. Each pipe contains 12 straight (horizontal) sections and 11 U-bends. The numeration of straight sections (horizontal rows) for each pipe, \( n \), starts from bottom to top (figure 1c).

The non-dimensional horizontal spacing \( a = 2.61 \) (\( a = S_1/D \), \( S_1 \) being the transverse pitch). The chosen U-bend radius provided the non-dimensional vertical spacing \( b = 1.6 \) (\( b = S_2/D \), \( S_2 \) being the longitudinal pitch). The straight section length \( L_{\text{hor}} = 32.5 \cdot D \). The pipes were attached to the inlet and outlet collectors of \( D_{\text{col}} = 2.5 \cdot D \) with the 90°-bends. The height of the tube bank, i.e. the distance between the collectors, is \( H = 40 \cdot D \).

Figure 1a partially illustrates the computational domain adopted for the problem considered. The external boundaries of the computational domain in \( x \)-, \( z \)-, and \( y \)-directions were located at the distance of \( 50 \cdot D \) from the tube bank (not shown in figure 1). Previous computations resulted in a conclusion that this distance is sufficient to provide a domain-independent solution [5].

Figure 1. The computational domain: a) a 3D view near the pipes; A-A, B-B – sections for postprocessing; b) section \( x = \) const and c) section \( y = \) const at the middle of pipe III

The plain rectangular walls parallel to the \( x \)- and \( y \)- coordinate plane were taken as bounding elements of the tube bank (figure 1 illustrates the bounding walls locations). The dimensions of \( x \)-plane walls are \( L_{\text{w},x} \times H_w \), and those of \( y \)-plane walls – \( L_{\text{w},y} \times H_w \), where \( L_{\text{w},x} = 14 \cdot D \), \( L_{\text{w},y} = 45 \cdot D \), \( H_w = 51 \cdot D \).

Three cases with various bounding walls arrangement were considered: Case #1 when the tube bank is bounded by four \( x \)- and \( y \)-plane walls (i.e. from all vertical sides; the computational domain for this case is shown in figure 1a), Case #2 – by two \( y \)-plane walls only, and Case #3 – by two \( x \)-plane walls only. In addition, Case #4 without any bounding wall was considered.

2.2. Turbulence modelling and water flow parameters
Sea water was assumed as an incompressible fluid with constant physical properties taken at 25°C. Density \( \rho = 997.1 \text{ kg/m}^3 \), viscosity \( \mu = 8.9 \times 10^{-4} \text{ kg/m/s} \), thermal conductivity \( \lambda = 0.611 \text{ W/m-K} \), and specific heat \( c = 4180.9 \text{ J/kg-K} \). The correspondent Prandtl number \( Pr = \mu c/\lambda = 6.09 \).
The no-slip conditions were set at the tube bank and collector walls. For the bounding tube bank walls the slip conditions (with zero shear stresses) were used. Far from the cooler at the external boundaries the uniform pressure was assumed. The $T_0 = 20^\circ$C value of far-field water temperature was set at these external boundaries. This temperature value corresponded to the typical conditions for a test (pilot) system in summer. The uniform temperature was set at the tube bank walls as $T_w = 30^\circ$C; acceptability and restrictions of this approximation were discussed in [6]. The Grashof number based on $D$ and on the reference temperature difference $\Delta T_{ref} = T_w - T_0$ was $Gr = 2\times10^5$.

The turbulent water flow was simulated using the full unsteady Navier-Stokes equations with no turbulence model and the energy equation. The buoyancy effects were represented by the Boussinesq approximation. The thermal expansion coefficient was set as $\beta_T = 2.057\times10^{-4}$ 1/K.

2.3. Computational aspects and solver settings

The computational grids were generated in the Gambit mesh-generator. The hybrid grid of about 44.3 million cells was used in the calculations for all four cases. For the external water flow domain, in the boundary layer near tube bank walls, and everywhere between the straight sections of the pipe the mesh consisted of hexahedral mesh elements. 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 was equal to 0.02 mm.

Numerical simulation was performed using ANSYS Fluent CFD-package, version 19.1. The second-order spatial discretization option (the second order upwind scheme, SOU) was chosen. The non-iterative time advancement scheme (NITA) based on the fractional step method was used. The value of the time step, $\Delta t$, was equal to 0.02 seconds. The length of time samples computed for statistics accumulation was about 1200 seconds (60 000 time steps) for each case. The averaging time was proved to suffice for obtaining statistically steady data.

The massively parallelized computations (up to 140 cores) were performed using the computational resources of the SPbPU supercomputer center (http://scc.spbstu.ru/).

3. Results and discussion

3.1. Draft water flow formation

In the region near the tube bank the draft water flow forms due to the buoyancy effects. Figures 2, 3 show time-averaged $z$-velocity distributions and pathline plots for four cases considered. Table 1 gives time-averaged values of water mass flow rate penetrating to the inter-tube space from different sides of tube bank. Notation in the table is in accordance with the coordinate system chosen (figure 1a): $G_x$ is the mass flow rate from two $x$-plane sides of the tube bank (from U-bends), $G_y$ – from $y$-plane sides, and $G_z$ – from $z$-plane sides. $G_{z_bot}$ is at the section below the tube bank; $G_{z_top}$ – above the tube bank.

For Case #1 the tube bank is surrounded by bounding walls from all vertical sides, therefore buoyancy effects initiate a flow field that looks like a channel flow with a constant mass flow rate (through the channel with the dimensions $L_w_x \times L_w_y \times H_w$). In this case, the streamlines are almost parallel to each other and to $z$-axis. Velocity fields inside the tube bank do not depend on the vertical section position (compare, e.g., fields at cross-sections A-A and B-B in figure 2a); visible spatial non-uniformity is due to the local effects at the entrance region and near the pipes (see, e.g., figure 3a).

The flow structure changes substantially at the side flow from the $x$- and/or $y$-direction (Cases #2, #3, and #4). As could be seen from figure 2b-c, the flow patterns at the $x$- and $y$-planes of the tube bank depend much on the section position that points to strong non-uniformity in the horizontal directions. For Cases #2, #3 and #4 with the side flow, the values of the draft velocity at the side cross-section B-B are much lower than the values at the middle section A-A.

According to table 1, with the absence of the side walls the water flow to the inter-tube space is mainly from the sides of the tube bank. The mass flow rate at the section above the tube bank does not depend much on the case considered (about 3 kg/s). Opposite to that, for Cases #2, #3 and #4 the mass flow rate below the tube bank is several times lower than the mass flow rate above the tube bank.
Table 1. Time-averaged mass flow rate at the boundaries of the tube bank, kg/s.

| Mass flow rate, kg/s | Case #1 | Case #2 | Case #3 | Case #4 |
|----------------------|---------|---------|---------|---------|
| $G_x$                | -       | 1.76    | -       | 0.54    |
| $G_y$                | -       | -       | 2.35    | 1.73    |
| $G_{z\text{,bot}}$  | 3.06    | 1.25    | 0.94    | 0.80    |
| $G_{z\text{,top}}$  | 3.07    | 3.05    | 3.26    | 3.08    |

Figure 2. Fields of $z$-velocity at the cross-sections A-A and B-B for Cases: a) #1, b) #2, c) #3, d) #4.

Figure 3. Fields of $z$-velocity at the cross-section C-C for Cases: a) #1, b) #2, c) #3, d) #4.

Figure 4 shows the mean draft velocity and the bulk water temperature evolution with the pipe straight section number for the middle pipe (III) and for the side pipe (I) of tube bank (to get the mean values, time- and area-averaging over the horizontal sections located at the distance $1.1D$ below the pipe straight section was performed, see [7] for details). The values of the draft velocity are varied in the range from 0.5 cm/s under the 1st straight section to 2 cm/s (side pipe) or even 3 cm/s (middle pipe) under the 12th straight section. For both velocity and temperature values, strong variation along and across the tube bank is detected for the cases with side flow (#2, #3, and #4).
Figure 4. Time- and area-averaged values of a,b) z-velocity and c,d) temperature at horizontal cross-sections located at the distance of 1.1D below each straight section of a,c) side and b,d) middle pipe.

3.2. Heat Transfer Evaluation

Figure 5 gives the information on the mean Nusselt number for all cases considered. The Nusselt number is defined as $Nu = q_w D/\lambda \Delta T_{ref}$, where $q_w$ is the heat flux area-averaged over the surface of the correspondent straight section. Time evolution of the area-averaged Nusselt number shown in figure 5a for Case #1 illustrates high level of flow unsteadiness at the upper part of the tube bank (at $n = 7$ and $n = 12$); the chaotic pulsations with the amplitude of about 10% are detected even though area-averaging is performed. Other three plots in figure 5 give $<Nu>$ values, i.e. time-averaged $Nu$.

Figure 5. a) Time evolution of $Nu$ area-averaged over straight sections $n = 1, 5, 7, 12$; middle pipe III is considered for Case #1; b,c) time- and area-averaged $Nu$ for b) middle pipe III and c) side pipe I; d) same as b,c), but for each $n$ data are additionally averaged over six straight sections.
Table 2. Integral heat output from the tube bank.

| Case | #1  | #2  | #3  | #4  |
|------|-----|-----|-----|-----|
| Q, kW| 23.5| 21  | 21.1| 22.2|

Time-averaged $Nu$ vs. straight section number plots are given in figure 5b for the middle pipe III and in figure 5c for the side pipe I. For all four cases the values of $Nu$ increase along the height of tube bank. The rapid growth in $Nu$ is observed at the 7th straight section of the tube bank that demonstrates the transition from almost laminar (steady-state or periodic) to close-to-turbulent (chaotic) regime; this transition is visible also in the velocity distributions discussed above. It is remarkable that if all four side walls are installed, $Nu$ is relatively high at the initial straight sections due to high velocities there. The effect of installing two walls only is not pronounced, especially for the side pipe. Additional averaging over six straight sections at the same height illustrated in figure 5d demonstrates that the total side flow effect on the cooler heat transfer is not large. A comparison of the total heat output from the tube bank, $Q$, given in table 2 also proves low sensitivity of integral characteristics to the wall installation. Thus, the cooler effectiveness does not depend much on arrangement of various bounding elements that could be helpful on the compressor unit design stage.

4. Conclusion

The study considered a model of a passive cooler based on the buoyancy effects. The focus was on the 3D unsteady external buoyancy-induced water flow through a staggered tube bank at moderate Grashof number. Full Navier-Stokes equations with no turbulence model were solved numerically.

The parametric computations were performed to investigate the effect of bounding walls on inter-tube flow structure and heat transfer. Strong side flow influence on the local characteristics has been found. The effect of the bounding elements on the integral characteristics is less pronounced: the only exception is the case with four side walls installed simultaneously and producing slight chimney effect that leads to draft flow acceleration and heat output growth. The results of the parametric study allow concluding that the cooler effectiveness does not depend much on the bounding elements arrangement.

Acknowledgments

The work was supported by the Russian Foundation for Basic Research (grant No. 18-08-00669).

References

[1] Fantoft R 2005 Subsea gas compression – challenges and solutions Offshore Technology Conference, 2-5 May 2005, Houston, Texas (OTC paper № 17399) pp 1-6
[2] Kleynhans G, Brenne L, Kibsgaard S and Denti P 2016 Development and Qualification of a Subsea Compressor Offshore Technology Conference, 2-5 May 2016, Houston, Texas (OTC paper № 27160-MS) pp 1-38
[3] Gyles B R, Haegland B, Dahl T B, Sanchis A, Grafsonningen 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
[4] Leahy M, Jagannatha D, Chauvet C and Holbeach 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.
[5] Ivanov N G, Ris V V and Tschur N A 2012 Numerical Modeling of Side Flow Effect on Free Convection Heat Transfer from an Aligned Plain-Tube Bank Teplovoe processy v tehnikе [Thermal processes in technology] 4(10) pp 434-441
[6] Zasimova M A, Ivanov N G, Ris V V and Tschur N A 2018 High Temperature, 56 (5) 711
[7] Ivanov N, Ris V, Tschur N and Zasimova M 2019 Computational Thermal Sciences, 11 (1-2) 1