Investigation of Impact Jets Flow in Heat Sink Device of Closed-Circuit Cooling Systems

D A Tokarev¹, V V Yenivatov¹, S S Sokolov², V L Erofeev²

¹ Sevastopol State University, 33 University St., 299053, Sevastopol, Russia
² Admiral Makarov State University of Maritime and Inland Shipping, 5/7, Dvinskaya St., Saint-Petersburg, 198035, Russia

E-mail: sergiiblack@gmail.com

Abstract. The flow simulations of impact jets in the heat sink device of the closed-circuit cooling systems are presented. The analysis of the rate of fluid flow in the heat sink device with the jet supply coolant is given.

1. Introduction
An important problem in the modern shipbuilding industry is to improve the power plants on ships, associated with increased reliability, efficiency and environmental safety of operation. To improve the characteristics of such power plants, as well as to reduce operating costs and a negative impact on the marine ecosystem, closed-circuit cooling systems are introduced in the practice of shipbuilding. A feature of the closed-circuit cooling systems is the refusal to take seawater into the system, but specially prepared fresh water circulates in this closed-circuit system. Depending on the type of vessel, a heat sink in the sea water may be accomplished by means of ship cladding heat exchangers with spray coolant supply to the heat exchange surface [1].

2. Analysis of research and publications
The motion study concerning the impact of the jets has been widely discussed [2-6]; at the same time, the various ways of organizing the fluid flow were considered. In [4], the results of research flow of cooled liquid to the heat exchange surface in the cladding of marine heat exchangers were presented.

The solution of the Navier-Stokes equations can be obtained by means of a set of mathematical, physical and numerical methods which form the computational fluid dynamics [5]. There are more than a dozen software systems to solve this problem numerically, as well as a large number of narrow profile programs. Narrow profile programs are designed to solve a specific problem privately. Due to the large number of individual tasks, it is difficult to determine the exact number of narrow-profile programs.

3. Statement of the base material
Reasoning from theoretical premises, the main consequence of nanopowder introduction into the melt should be refinement of the macro- and microstructure, as the powder particles must serve as nuclei of new grains. Figures 1 and 2 show photos of the microstructure of cast samples from the bronze of the lead-tin bronze grade, both modified by SDP of aluminium oxide and without modifier addition.

Content from this work may be used under the terms of the Creative Commons Attribution 3.0 licence. Any further distribution of this work must maintain attribution to the author(s) and the title of the work, journal citation and DOI.
The aim of the article is to present the current results of the simulation of impact of the jets in the heat removal devices for outboard water in the closed cooling systems of power plants of ships and marine hardware.

Fig. 1 shows the scheme of the jet flow in heat exchangers of impact with the destruction of the boundary layer, wherein the limits of the hull framing movements are not taken into account.

Impact jet is formed at the exit of the channel through which the coolant is supplied. Jet formation character is defined by the distance from the tube to the inner surface of the heat exchanger, the fluid flow rate, diameter channels, geometric parameters of the heat sink device.

Calculation of the emerging jet profile is possible analytically when using the Navier-Stokes equations. Thus it is necessary to introduce the use of the assumption of a viscous Newtonian fluid. Let us write the system of partial differential equations:

\[
\frac{\partial \vec{v}}{\partial t} = \left( \vec{v} \cdot \nabla \right) \vec{v} + \nu \Delta \vec{v} - \frac{1}{\rho} \nabla p + \vec{f} \tag{1}
\]

where \( t \) – the time, \( \nu \) – the kinematic viscosity, \( \rho \) – density, \( p \) – pressure, \( \vec{v} = (v_1, v_2, \ldots, v_n) \) – velocity vector field, \( \vec{f} \) – vector field of mass forces, \( \nabla \) – operator nabla, \( \Delta \) – vector Laplace operator.

Since both \( p \) and \( \vec{v} \) are functions of time and coordinates of the region in which the fluid moves, valid entry is \( \rho(t, x), \vec{v}(t, x), x \in \Omega \), where \( \Omega \subset \mathbb{R}^3 \).

The system of equations (1) describes the motion of the fluid in compliance with the conditions of continuity \( \nabla \cdot \vec{v} = 0 \), boundary conditions \( \vec{v} |_{\partial \Omega} = 0 \) and initial conditions \( \vec{v} |_{t=0} = \vec{v}_0 \).

Figure 1. The scheme of the jet flow in heat exchangers of impact with the destruction of the boundary layer: \( v \) - flow velocity at the entrance to the channel; \( d \) - diameter of the channel; \( L \) - distance from the outlet channel to the heat transfer surface.

The system of equations (1) in a closed form is a complex task that depends on the initial and boundary conditions. In solving the system of equations (1), the analytical method resorted to a number of assumptions and simplifications, which in some cases gives a particular solution. One of the ways to obtain an approximate solution of the system is the use of numerical methods [6-11]. The first approach is justified in dealing with non-standard specific tasks. In other cases, it is advisable to use specialized software designed for the numerical solution of the Navier-Stokes equations.
When solving this problem, the following algorithm was used: the basic parameters of media, boundary conditions, initial conditions, and other simulation conditions were set. The calculation in the software environment was made, results were obtained; the analysis of the results is performed.

Fig. 3 shows the results of simulation of the velocity field in the experimental stand with color-coded velocity of fluid flow. The diagram shows the emergence of vortices, which should positively affect the amount of adjacent spot (Figure 3.a and Figure 4). Previously, it has been found that there is the occurrence of secondary flows and liquid leak from the environment. It should be clarified that the rate of entrained liquid reaches 0.06 m/s (Fig. 4).

Fig. 5.a shows the velocity distribution with a maximum speed limit to 0.2 m/s: red colour corresponds to a maximum speed of 0.2 m/s, and green - 0.1 m/s. In the outlet ports the turbulent flow zone is observed, which is caused by the interaction of the secondary flow described above. The diameter of the jet hitting spots while maintaining the speed of 0.2 m/s is 0.23 m when speeds are preset.
Fig. 5.b shows the jet hitting the spot. The shape approximates to a circle, indicating the presence of computational errors, since the results of imaging studies determined the spot shape as the form of concentric circles. The results of this simulation of impact jet show good results with the above disadvantages associated with the imposed restrictions and simplifications.

The number of jets should be enlarged to increase the area of the inner surface of the intensified heat transfer. The calculation shows the lack of viability of this method: the spot size at a rate is not lower than 0.1 m/s depending on the liquid flow rate of the speed at the tube inlet. Increasing the number of tubes reduces the flow through any of them respectively. As shown above, the spot diameter at a high speed is 0.23 m and the area of the spot is 0.042 m$^2$; the liquid volume flow is a function of fluid velocity at the inlet of the cylindrical tube, $7.85 \times 10^3$ m$^3$/s.
Figure 7. The results of the fluid velocity imitation modeling

Figure 8. The results of the fluid velocity modeling ($L = 7 \cdot 10^{-3}$ m; grid spacing is $5 \cdot 10^{-3}$ m, displayed fluid flow rate is limited to 0.2 m/s): a) a side view; b) a front view of the part of the heat exchange surface

4. Conclusions
Presented flow simulations of impact of the jets in the heat sink device in the closed-circuit sea water cooling systems showed good agreement with the results of earlier studies, which clearly confirms the prospects of this method of intensification of heat removal.

The development of the model is of a certain interest which allows analysing flow velocity in the heat sink devices with spray coolant supply with installation of a modified form of the screen, as well as clarification of the existing settlement dependencies underlying the relevant procedures.

Calculation methods implementation of closed-circuit cooling systems will allow using a heat sink area more effectively, reducing operating costs and improving energy performance, thereby ensuring the more widespread use of reliable and environmentally friendly cooling systems of power plants of ships and marine hardware.

5. Acknowledgments
The research was performed in the research laboratory "Effective power plants and technologies" of Federal State Autonomous Educational Institution of Higher Education "Sevastopol State University".

References
[1] Fedorov K Yu 2012 Heat transfer (Sevastopol national technical University Press)
[2] Middelberg G, Herwig H, Convective Heat Transfer under Unsteady Impinging Jets: The Effect of the Shape of the Unsteadiness, Heat Mass Transfer, 45, 2009, 1519-1532
[3] Yudaev B N, Mikhailov M S, Savin V K 1977 Heat transfer in jet interaction with barriers Engineering 248
[4] Fedorov K Yu 1991 Devices and cooling systems of power plants marine hardware (Sevastopol)
[5] Roach P 1980 Computational Fluid Dynamics (Mir)
[6] Chernyi S 2016 Transport and Telecommunication Journal 17 3 DOI: 10.1515/ttj-2016-0018
[7] Sedova N A, Sedov V A, Glushkov S V 2016 Vibroengineering PROCEDIA 8 506-511
[8] Budnik V, Chernyi S 2016 Procedia Engineering 150 2150-2156
[9] Chernyi S 2016 Neftyanoe Khozyaystvo - Oil Industry 2 106-110
[10] Nyrkov A, Shurenko A, Sokolov S, Chernyi S and Korotkov V 2017 International Journal of Electrical and Computer Engineering (IJECE). 7 (6) 3578
[11] Nyrkov A, Chernyi S, Zhilenkov A and Sokolov S 2016 Proceedings of the 26th International DAAAM Symposium 2016, 0672-0677