Study of possibility of using axisymmetric body as heat sink for cooling machine parts

A Palii¹,², I Tolmacheva¹, T Novoselova¹, and J Akopdzhanyan¹

¹Polytechnic Institute (branch) of DSTU in Taganrog, st. Petrovskaya 109A, 347904, Taganrog, Russia
E-mail: a.v_palii@mail.ru

Abstract. The paper presents the results of the study of possibility using a body with an optimized shape with a minimum aerodynamic drag force as a heat sink in a convective gas flow for cooling machine parts. The relevance of cooling or heat sink systems for mechanical engineering is not in doubt. Maintaining the optimum thermal state of machine parts within the specified limits is necessary to prevent heating, which can cause violations of normal operating conditions, and, as a result, increased wear, jamming and breakage of parts. To ensure the normal operation of the system, it is necessary to cool the parts contacting the hot gases, removing heat from them to the atmosphere directly, or using an intermediate body (water, low-freezing liquid). Heat sink is an aerodynamic process, a process of heat mass transfer of a substance, and aerodynamic characteristics and, in particular, the nature of streamlining of bodies of the simplest forms are of great scientific and practical interest. Bodies of complex shape can always be represented as a combination of simpler ones, for which it is easy to explore and calculate the flow paths, on the basis of which analytical methods for calculating aerodynamic characteristics are compiled. A computational experiment was conducted in the software product Ansys Fluent. The conditions of the experiment (comparison of heat removal bodies) during numerical modeling are the preservation of constants: the volume and shape of the working zone; distances from the sources, drains and centers of bodies; aerodynamic flow rates; masses of compared bodies; thermal power of the source and other parameters in addition to the shape of the surface. The resulting optimized shape of the body coincides with the current lines, which is the main advantage, since no separation from the surface is observed when streamlining the flow. Thus, the entire surface area will be the effective surface area of the heatsink as opposed to other known shapes of bodies, whereby the temperature of the heat loaded element placed in the center of the heatsink will decrease.

1. Introduction
Maintaining the optimum thermal state of machine parts within the specified limits is necessary to prevent heating, which can cause violations of normal operating conditions, and, as a result, increased wear, jamming and breakage of parts. To ensure the normal operation of the system, it is necessary to cool the parts contacting the hot gases, removing heat from them to the atmosphere directly, or using an intermediate body (water, low-freezing liquid). Despite the apparent simplicity of the external shapes, the streamlining of simple bodies is a very complex process even in isolated conditions. Even more difficult is the streamlining and calculation of the aerodynamic characteristics of bodies, when their simplest geometric shapes are part of even more complex shapes. In this case, the mutual influence of individual areas of the surface of the body on each other is manifested, significantly
complicating the original picture currents. At the same time, the number of geometric parameters affecting the flow structure increases. [1-4].

Aerodynamic characteristics and in particular the nature of the streamlining of bodies of the simplest forms represent a great scientific and practical interest. Bodies of complex shape can always be represented as a combination of simpler ones, for which it is easy to explore and calculate the flow paths, on the basis of which analytical methods for calculating aerodynamic characteristics are compiled. The initial data for this are information obtained from experiments, including a computational experiment with bodies of the simplest forms [5,6].

In this work, numerical modeling is carried out to confirm the theoretical solution of the problem of finding the shape of an optimal aerodynamic body with minimal aerodynamic drag, which is reduced to the calculation of the mathematical formula of the curve (current line) forming this body by rotation about an axis coinciding with the flow direction of a given speed, which we obtained in our work earlier [7].

2. Setting the task

We have previously optimized the shape of the body in the gas stream in order to minimize its aerodynamic drag. This shape was obtained by rotating the profile (curve) about an axis coinciding with the flow direction of a given speed. The mathematical formula of the curve is described by the following expression [8-10]:

\[
1 - \frac{a^2}{(x^2 + y^2)} \quad y = \text{const},
\]

where \(a\) – is the radius of the circle according to the magnitude of the moment of the dipole.

We determine by applying a homogeneous flow parallel to the OX axis at speed \(v\) to the speed field of the dipole with a positive moment, which corresponds to the flow of gas from the dipole towards the incoming homogeneous flow (Figure 1).

![Figure 1](image)

**Figure 1.** Combined current lines when homogeneous flow is superimposed on the dipole high-speed field with positive moment.

This article considers the possibility of using the described body with an optimized shape with a minimum aerodynamic drag force as a heat sink in a convective gas flow.

When a uniform flow parallel to the axis is superimposed OX at speed \(|v_\infty|\) and complex potential:

\[ Z_1 = |v_\infty|z, \quad (1) \]
on high-speed field of dipole with complex potential:

\[ \chi_2 = m/2\pi z, \]  
(2)

and positive moment \((m > 0)\), which corresponds to the flow of liquid from the dipole towards the oncoming flow, we get a complex potential:

\[ \chi = \chi_1 + \chi_2 = |v_\pi|z + (m/2\pi)(1/z). \]  
(3)

By highlighting the imaginary part in this expression, we define the current function \(\Psi\) as

\[ \Psi = |v_\pi|y - (m/2\pi)(y/(x^2 + y^2)). \]  
(4)

For the zero line of current \(\Psi = 0\), we get the equation:

\[ \left[|v_\pi| - (m/2\pi)(1/(x^2 + y^2))\right]y = 0. \]  
(5)

Selecting the dipole moment value equal to:

\[ m = 2\pi a^2|v_\pi|, \]  
(6)

get a zero current line in the form of a radius circle with a center at the origin and the OX axis.

3. Description of the computational experiment

In order to now move from the aerodynamic task to the thermal task, it is necessary to obtain equivalent temperature surfaces corresponding to the optimized shape of the heat sink in the convective flow of a given speed and a given speed of thermal energy propagation from the point heat source. To do this, we replace the spherical region of zero potential of the dipole field corresponding to a similar spherical region from a point heat source of double power. The general nature of the combined current lines will not change, but the combined temperature lines (surfaces) from the heat source in the convective flow will be obtained [11-17].

The main advantage of the resulting optimized shape of the body is that it coincides with the current lines, thereby, when it flows around, the separation of flow from the surface will not be observed. Thus, the entire surface area will be the effective surface area of the heatsink, whereby the temperature of the heat loaded element placed in the center of the heatsink will be reduced.

To prove this statement, we give the results of a numerical simulation carried out in the Ansys Fluent software product on the study of the temperature of a point heat-loaded element in a heat sink of an optimized shape, calculated for a given convective flow rate and, for example, a ball, which is notable for the minimum surface area at equal body volumes, distances from the sources, drains and centers of bodies; gas flow rates; mass of bodies, thermal power of the source and other parameters in addition to the shape of the surface (Figure 2).
Figure 2. Temperature field distribution in ball (a) and optimized axisymmetric (b) heatsinks.

The figure shows that the temperature of the point source in the center of the optimized heat sink (for a given speed - spindle-shaped) is lower than in the ball. From this distribution of the temperature field, it can be erroneously concluded that a decrease in temperature occurs due to a larger surface area. It should be noted here that not the total surface area of the heatsink is important, but an effective one, that is, taking part in the removal of heat by the flow without its separation. Figure 3 shows that the separation of the flow from the ball occurs approximately at the border of the left and right hemispheres, that is, the right part of the heat sink is practically not streamlined by the flow. Separation from the optimized heatsink does not occur, therefore, the entire surface area is involved, since the heatsink is made along equivalent temperature lines (equipotential surfaces).

Changing the conditions of the experiment by changing the flow rate at which the former spindle shape will no longer be optimal, we will get a significant increase in the temperature of the source compared to the source in the spherical heat sink.

4. Main findings
After changing the numerical simulation conditions, the source temperature in the ball heatsink changed slightly, and the source temperature in the spindle significantly increased. This is due to the fact that the zone of separation of the flow from the ball heatsink practically does not change, and from the spindle-shaped it moves significantly closer to the source. Thus, although the total area of the heat sink is maintained, the effective area is substantially reduced, and therefore the temperature of the heat loaded element increases.
Ultimately, the efficiency of the heat sink may depend on a variety of factors such as the nonlinearity of the flow, the ratio of flow rates and thermal energy propagation from the source, surface roughness, color, etc. The equilibrium between the front and rear of the heat sink surface may be questioned. But the main unchanged rule for obtaining an effective form should remain the execution of it on equitemized surfaces from a source in a convective flow of a given speed.

References

[1] Houghton E L, Carpenter P W, Collicott S H, Valentine D T 2013 Aerodynamics for Engineering Students, Elsevier Ltd, pp 717

[2] Townsend A 2013 A graduate introduction to numerical methods: From the Viewpoint of Backward Error Analysis. Springer, New York, Heidelberg, pp 252

[3] Vetchanin E V, Mamaev I S, Tenenev V A 2013 Regular and Chaotic Dynamics Vol 18 Pp 100-117

[4] Pradhan S 2017 The generalized Onsager model for a binary gas mixture with swirling feed Proc. 8th AIAA Theoretical Fluid Mechanics Conf. (Denver; United States/AIAA) pp 214–220

[5] Yin Z, Gao C and Yang X 2017 Numerical study of hydrodynamic behavior of heaving cylinder buoy under regular wave Taiyangneng Xuebao Acta Energiae Solaris Sinica Vol 38(12) 3363–7

[6] Gorskiy V V and Sysenko V A 2014 Modeling of gas flow through laminar boundary debris on the surface of the hemisphere in supersonic air flow Mathematical modeling and numerical methods Vol 4(4) 88-94

[7] Chernov N N, Palii A V, Saenko A V and Maevskii A M 2018 A method of body shape optimization for decreasing the aerodynamic drag force in gas flow Technical Physics Letters Vol 4 328–30

[8] Viola P and Jones M J 2014 Robust real-time face detection International Journal of Computer Vision Vol 57 137–54

[9] Ignatyev V, Soloviev V, Beloglazov D, Kureychik V, Kovalev A and Ignatyeva A 2019 The fuzzy rule base automatic optimization method of intelligent controllers for technical objects using fuzzy clustering Proc. Creativity in Intelligent Technologies and Data Science Third Conf., CIT&DS(Volgograd, Russia) 135–52

[10] Ignatyev V, Soloviev V, Beloglazov D, Kureychik V, Ignatyeva A and Vorotova A 2019 System for automatic adjustment of intelligent controller parameters Proc. Creativity in Intelligent Technologies and Data Science Third Conf., CIT&DS(Volgograd, Russia) 226–42

[11] Mokhtari M, Permoon M R, Haddadpour H 2019 Aeroelastic analysis of sandwich cylinder with fractional viscoelastic core described by Zener model. Journal of Fluids and Structures Vol 85, 1–16

[12] Ziemkiewicz D Simple analytic equation for airfoil shape description

[13] He S, He K, Seddighi M 2016 Laminarisation of flow at low Reynolds number due to streamwise body force. Journal of Fluid Mechanics Vol 809, 31–71

[14] Moallemi M, Clifford J, Neighbors J, Pesce J, Towhidnejad M 2016 Flight dynamics and surveillance equipment simulation for trajectory based operation in unmanned aerial systems. In: AIAA/IEEE Digital Avionics Systems Conference – Proceedings, art. 7778002

[15] Mint Z M, Khlopov Ayu 2010 Aerodynamic characteristics of aircraft hardware of complex shape taking into account potential interaction of molecular flux with the surface. Scientists of note TSAGI Vol 41(5), 33-45

[16] Faraponov V V, Savkin N V, Dyachevsky A S, Chupashev A V 2013 Aerodynamics of bodies with small elongation in subsonic and near-sound modes of flow. News Physics Vol 56(6-3), 77-79

[17] Liu J, Hou Z, Chen X, Gao X 2012 Numerical simulation on the aerodynamic performance of hypersonic glide vehicle. Guofang Keji Daxue Xuebao. Journal of National University of Defense Technology Vol 34(4), 22–27