This study investigates the sequential and continuous formation of thermal fields in the main fairing of a launch vehicle when using protective screens. While thermostating, it is necessary to predict the risk in overheating the payload body and, if necessary, take measures to reduce the temperature near the payload.

An engineering solution to this problem can be found through the use of protective screens of various configurations inside the main fairing. These screens reduce the heat flow from the heated outer wall of the fairing to the payload surface. However, there are no standard methods for solving this problem.

To evaluate the effectiveness of this protection, a numerical model based on the fundamental equations of continuum mechanics has been constructed. The modeling equations include the energy equation and the equation of motion of a non-viscous gas. Using the numerical model built, a computational experiment was conducted, which confirmed the effectiveness of using protective screens to shield the payload body from excessive heating. The computer time required to perform the computational experiment is 3 seconds. This makes it possible to perform a significant number of calculations in a working day.

The proposed simple technical means for protecting the payload from excessive heating could be used in the design of new models for rocket technology. Applying these screens slightly reduces the need for large volumes of clean air. The numerical model built could be used at specialized organizations at the "for-sketch" design stage. Numerical experiments have shown that the use of protective screens inside the main fairing makes it possible to achieve a temperature 2–4°C lower than the maximum permissible temperature near the payload

Keywords: thermal pollution, main fairing, protective screen, numerical model, computational experiment

UDC 519.6:629.76

DOI: 10.15587/1729-4061.2025.347454

CONSTRUCTION OF A MATHEMATICAL MODEL OF THE HEAT AND MASS TRANSFER PROCESS IN THE MAIN FAIRING OF A LAUNCH VEHICLE AT THE PRE-LAUNCH PREPARATION STAGE

Mykola Biliaiev
Corresponding author
Doctor of Technical Sciences, Professor
Department of Hydraulics, Water Supply and Physics*
E-mail: diit.hydro.eco@gmail.com

Viktoriia Biliaieva
Doctor of Technical Sciences, Professor
Department of Energy Systems and Energy Management*

Tetiana Rusakova
Doctor of Technical Sciences, Professor**

Vitalii Kozachyna
PhD, Associate Professor
Department of Hydraulics, Water Supply and Physics*

Pavlo Semenenko
PhD
Department of Aerogas Dynamics and Heat and Mass Transfer
Yuzhnoye State Design Office
Krivorozhskaya str., 3, Dnipro, Ukraine, 49008

Oleksandr Berlov
PhD, Associate Professor
Department of Labor Protection, Civil and Technogenic Safety*

Pavlo Kirichenko
PhD, Associate Professor
Department of Heat, Gas, Water Supply, Water Drainage and Ventilation
Kryvyi Rih National University
Vitaliya Matusevicha str., 11, Kryvyi Rih, Ukraine, 50027

Nataliia Hrudkina
Doctor of Technical Sciences, Associate Professor
Department of Natural Sciences and General Engineering Disciplines
Technical University "Metinvest Polytechnic"
Pivdenne highway, 80, Zaporizhzhia, Ukraine, 69008

Yuliia Voitenko
PhD**

Olena Dolzhenkova
PhD**

*Ukrainian State University of Science and Technologies
Lazaryan str., 2, Dnipro, Ukraine, 49010
**Department of Life Safety
Oles Honchar Dnipro National University
Nauky ave., 72, Dnipro, Ukraine, 49010

Received 08.10.2025
Received in revised form 25.11.2025
Accepted 18.12.2025
Published 30.12.2025

How to Cite: *Biliaiev, M., Biliaieva, V., Rusakova, T., Kozachyna, V., Semenenko, P., Berlov, O., Kirichenko, P., Hrudkina, N., Voitenko, Y., Dolzhenkova, O. (2025). Construction of a mathematical model of the heat and mass transfer process in the main fairing of a launch vehicle at the pre-launch preparation stage. Eastern-European Journal of Enterprise Technologies, 6 (1 (138)), 71–79. https://doi.org/10.15587/1729-4061.2025.347454*

1. Introduction

Inside the main fairing of a launch vehicle, at the stage of pre-launch preparation, a microclimate is created to provide the necessary conditions for the payload. One of the main tasks is to ensure a certain temperature regime inside the fairing during forced air supply to it [1]. In this case, there is a need to induce a certain temperature field near the sensitive surfaces of the payload (these are antennas, repeaters, navigation sensors, solar panels, and optical devices). Some elements, such as batteries, must be in the temperature range from 0°C to 20°C [2]. At the same time, it is very often necessary to take into account the complex layout of spacecraft inside the main fairing. For example, the layout, according to the "tandem" scheme, is typical for launching groups of spacecraft (Fig. 1).

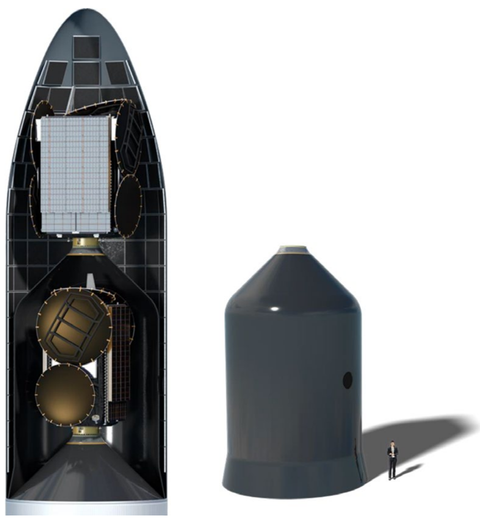

Fig. 1. Main fairing layout with tandem spacecraft arrangement (NewGlenn) [3]

When predicting temperature fields inside the main fairing, it is necessary to take into account the temperature of its outer walls. The wall of the main fairing is a thin-walled structure, prone to various hazards during pre-launch operations [4, 5]. The temperature on the outside of the fairing can vary in a wide temperature range from +40°C to –40°C as spaceports are located in different climatic zones. This temperature has a direct impact on the formation of the temperature regime inside the fairing, as well as near the sensitive surfaces of the payload.

Different approaches are used to predict temperature fields inside the fairing and ensure the desired temperature regime [6]. Uniform air supply of a certain temperature is not always effective, from the point of view of the formation of certain air velocities between the fairing and the spacecraft. To enable the desired temperature regime, it is possible to use a heat-screening coating both outside the fairing and inside the fairing. The device is a rigid cylindrical shell with special drainage holes. This is reflected in special, so-called "consumer guides", in which rocket manufacturers describe in detail all the conditions that are specially created only to meet the requirements for ensuring the operability of the spacecraft [7–9]. The principle of tandem arrangement of spacecraft is currently offered in the launch services of the Ariane 6 and NewGlenn launch vehicles. Such a separate arrangement of two spacecraft is provided by a special cylindrical device. However, this approach leads to an increase in the "non-usable" mass of the vehicle. Thus, the current task is to devise means that contribute to the formation of the necessary temperature regimes near the payload and do not lead to a significant increase in the mass of the payload or an increase in energy to maintain the required temperature value near sensitive surfaces.

Inducing a microclimate for the spacecraft at the stages of pre-launch preparation is mandatory. Given the fact that spacecraft are constantly evolving in their complexity, this is a relevant area of scientific research.

2. Literature review and problem statement

In [10], the process of forced convection in a rectangular housing with a constant heat source is studied. Heat-generating components that dissipate a heat flux of no more than 350 W/m^2 were modeled in the volume. The air volume was forcibly blown by an air flow that ensured interaction with heated surfaces. The studies were conducted using the commercial software Fluent©. It was shown that the fan flow rate plays an important role in cooling the housing, as well as the location of the holes. Comparison of the Nusselt numbers related to housings with different locations of holes provides invaluable information when designing the optimal layout of the installation. The paper very well shows the relevance of research work on finding design solutions to the problems of distributing thermal fields in a closed volume during forced ventilation. There are still unsolved tasks on finding design solutions to problems in a cylindrical body with a constant heat source.

In [11], an experimental study is conducted with modeling of forced convection. The work reports a classic version of the study – heating and cooling of a copper plate. The purpose of the experimental study was to investigate the characteristic cooling time depending on the average air flow velocity. It is noted that the most important variable for the cooling time is the plate thickness. Experimental dependences of the plate cooling process were obtained. On their basis, empirical functions were determined to describe the plate cooling process. These studies are the so-called reference experiments on the heating and cooling process. Owing to these experiments, we have empirical functions that make it possible to conduct applied research. The cited work does not study the interaction of gas-dynamic vortices and stagnant zones, which significantly affect the cooling or heating time of the surface.

In [12], an experimental study was conducted to determine the influence of the axial fan speed, outlet height, position, area, and aspect ratio of the ventilated housing on convection heat transfer. A closed volume with a heating element inside was modeled. Forced ventilation was performed to assess the influence of air velocity on heat transfer. Flow velocities were limited to 4 m/s, which is the speed limit near the surfaces under study. Different positions of the air outlet were modeled at different flow velocities. The experiment was performed for different housing aspect ratios at constant heat source power. The authors conclude that the temperature increase is small in dependence on the outlet area with a sufficiently large spread of the Nusselt number for given flow velocities. Not taken into account in the work is the presence of the so-called "aerodynamic shadow zones" or zones with increased flow velocity. This is always observed with forced ventilation. Not taking into account the influence of these zones is a certain drawback of the cited paper.

In [13], a new type of air cooling system with small dimensions and flexible layout is proposed, namely micro air ducts with pinhole-sized ventilation holes. The air cooling system is numerically modeled using ANSYS Fluent CFD software. Part of the thermal control system of the space scientific payload is designed. For that multifunctional experimental platform, "rigorous" requirements for the environmental temperature index for various experimental modules inside are defined. Numerical simulation of the internal airflow confirmed that, compared with the conventional method, the temperature field and flow field of the new air cooling method are more uniform, and heat sources located at the edge of the rack can also be cooled better. The disadvantage is that the results are obtained for pinholes. Such holes form a directed air flow onto the surface to be cooled.

In [14], temperature and velocity fields in the air volume of the main fairing were obtained, taking into account the placement of the spacecraft in it. For this purpose, the forced ventilation process was simulated in the ANSYS CFD environment. A number of safety measures were proposed, in particular, the regulation of the air conditioning interface inside the fairing partition. The use of parallel air conditioning units was simulated. Special tests proved that the complex of safety measures taken is effective. The designed flow velocity distribution system has become decisive in the efficiency of heat removal from heat-emitting surfaces. The work uses the well-known and high-quality ANSYS software product. The disadvantage of its use is the dependence on a large amount of time for calculating one option. It is also necessary to take into account the requirements for computer power. These two features are crucial when organizing work at the design stages.

In [15], a simulation of the flow velocity and thermal fields for a spacecraft placed in the air volume of the main fairing was carried out. A high-speed numerical model was built to predict thermal fields in the main fairing of a launch vehicle during its forced ventilation. The proposed numerical model makes it possible to take into account the main physical factors that affect the air velocity in the main fairing during forced ventilation. The developed computer program allows for computational calculations in a short time. The disadvantage of the numerical model is that it models only the influence of the fairing walls and the surfaces of spacecraft that are thermostated.

Our review of the related literature demonstrates that the relevance of research into heat and mass transfer processes applies both to them [11] and to practical tasks [10, 12–15]. All this allows us to assert that the processes of forced ventilation of spacecraft do not yet have a single optimal qualitative solution in general.

An effective solution to this problem is likely the use of special screens that affect the formation of thermal fields. However, the use of such an approach requires the presence of a numerical model, which makes it possible at the decision-making stage to assess the effectiveness of this method for regulating temperature fields inside the fairing. The importance of building a numerical model relates to the fact that the customer of payload prohibits measuring the temperature field in the main fairing. Therefore, the only method for assessing the effectiveness of screens on the formation of thermal fields near the payload is mathematical modeling. There is no constructed numerical model of spacecraft forced ventilation in the volume of the main fairing, taking into account protective structures.

3. The aim and objectives of the study

The purpose of our study is to build a mathematical model of heat transfer, which makes it possible to determine the effect of screens on reducing the temperature near the sensitive surfaces of the payload in the main fairing. Determining this effect makes it possible to assess the effectiveness of screens for protecting the payload from excessive overheating.

To achieve the goal, the following tasks were set:

– to break down a numerical model for calculating the air flow velocity field in the main fairing of the launch vehicle at the pre-launch preparation stage;

– to construct a numerical model for calculating the temperature field in the main fairing of the launch vehicle at the pre-launch preparation stage;

– to conduct computational experiments based on the numerical models built to study the effectiveness of using screens to protect against heating of sensitive surfaces of the payload in the main fairing.

4. The study materials and methods

The object of our study is the consistent and continuous formation of thermal fields in the main fairing of a launch vehicle when using protective screens.

The working hypothesis assumes that the location of a protective screen inside the main fairing makes it possible to change the thermal regime in it and protect special surfaces of the payload from overheating. The location of the payload according to the "tandem" scheme inside the fairing is under consideration (Fig. 2).

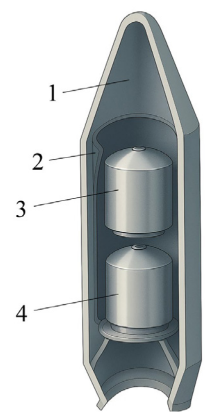

Fig. 2. Tandem payload arrangement in the main fairing: 1 — main fairing; 2 — protective screen; 3, 4 — payload

Within the framework of our working hypothesis, several engineering solutions are considered:

– scenario #1 – no protective screen in the fairing (basic version, Fig. 3);

– scenario #2 – long screen (Fig. 2, 4);

– scenario #3 – two short screens (Fig. 5);

– scenario #4 – two short screens + air blowing inside the fairing (Fig. 4).

To test the working hypothesis, a computational experiment was conducted based on the numerical models built. When constructing numerical models, the following assumptions were adopted:

– air viscosity does not affect the aerodynamics inside the fairing, only the diagonal components of the temperature transfer coefficient tensor are taken into account;

– the flow motion is vortex-free.

Theoretical study of thermal regime inside the main fairing with a protective screen is carried out by numerical integration of the modeling equations of energy and aerodynamics using the finite difference method.

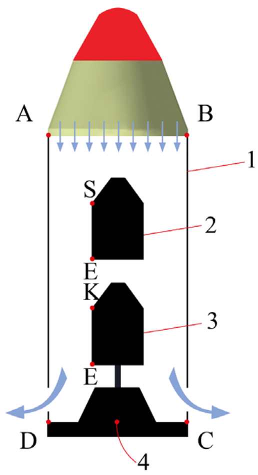

Fig. 3. Layout of the payload inside the fairing (no screen, scenario #1): 1 – fairing; 2, 3 – payload; 4 – adapter (ABCD – estimated area)

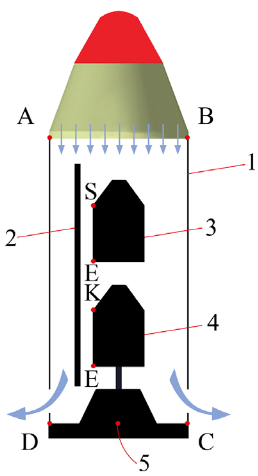

Fig. 4. Layout of a long screen inside the fairing (scenario #2): 1 – fairing; 2 – screen; 3, 4 – payload; 5 – adapter (ABCD – estimated area)

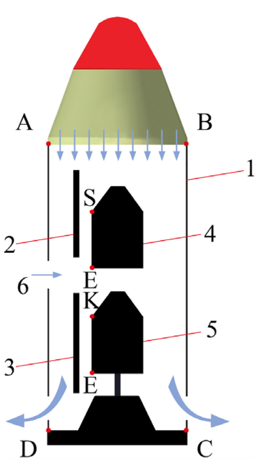

Fig. 5. Layout of short screens inside the fairing (scenario #3 and scenario #4): 1 – fairing; 2, 3 – short screen; 4, 5 – payload; 6 – air injection location (ABCD – estimated area)

Our proprietary software is used for numerical modeling: the computer program "START", developed in the FORTRAN programming environment.

5. Results of the study on constructing a mathematical model of the thermal regime inside the main fairing of a launch vehicle

5. 1. Construction of a two-dimensional numerical model for calculating the flow velocity field in the main fairing

Below is the procedure for building a numerical model to determine the flow velocity field in the main fairing at the stage of its thermostating. The scheme of the estimated area is shown in Fig. 3–5.

The estimated area in the main fairing is marked by the boundary *ABCD*, Fig. 3–5. To calculate the velocity field in the main fairing during its thermostating, the inviscid vortex-free flow model is used

$$\frac{\partial P^2}{\partial x^2} + \frac{\partial P^2}{\partial y^2} = 0, \quad (1)$$

where P is the velocity potential.

Boundary conditions for the aerodynamic equation are as follows:

– at the boundary A–B – the air flow enters the fairing at velocity U; the boundary condition takes the form $\partial P/\partial x = U$. When conducting a computational experiment, $U = \text{const}$ is assumed;

– at the air flow exit boundary: $P = P_0 + \text{const}$, where P_0 is an arbitrary constant. When performing the calculation, $P = 1000$ is assumed;

– at solid boundaries: $\partial P/\partial n = 0$, n is the unit vector of the external normal to the boundary.

If the velocity potential field is known, then the components of the air flow velocity vector are determined as follows

$$u = \frac{\partial P}{\partial x}, \quad v = \frac{\partial P}{\partial y}, \quad (2)$$

Numerical integration of equation (1) is carried out on a rectangular difference grid.

In advance, the Laplace equation (1) is reduced to the following form

$$\frac{\partial P}{\partial t} = \frac{\partial^2 P}{\partial x^2} + \frac{\partial^2 P}{\partial y^2}, \quad (3)$$

where t – fictitious time.

For numerical integration of equation (3), a two-step scheme for splitting the total approximation is performed. This scheme is as follows:

– step I

$$\frac{P_{i,j}^{n+\frac{1}{2}} - P_{i,j}^{n}}{\Delta t} = \left[\frac{-P_{i,j}^{n+\frac{1}{2}} + P_{i-1,j}^{n+\frac{1}{2}}}{\Delta x^2}\right] + \left[\frac{-P_{i,j}^{n+\frac{1}{2}} + P_{i,j-1}^{n+\frac{1}{2}}}{\Delta y^2}\right], \quad (4)$$

– step II

$$\frac{P_{i,j}^{n+1} - P_{i,j}^{n+\frac{1}{2}}}{\Delta t} = \left[\frac{P_{i+1,j}^{n+1} - P_{i,j}^{n+1}}{\Delta x^2}\right] + \left[\frac{P_{i,j+1}^{n+1} - P_{i,j}^{n+1}}{\Delta y^2}\right]. \quad (5)$$

Thus, to determine the field of function P (velocity potential), it is necessary to sequentially solve equations (4) and (5) in each difference cell (control element). The solving procedure ends when the condition is satisfied

$$\left|P_{i,j}^{n+1} - P_{i,j}^{n}\right| \le \varepsilon,$$

where ε is a small number (it is assumed that $\varepsilon = 0.001$); n is the iteration number.

After calculating the velocity potential field, the components of the air flow velocity vector are calculated based on the following dependences

$$u = \frac{P_{i+1,j} - P_{i,j}}{\Delta x}, \quad v = \frac{P_{i,j+1} - P_{i,j}}{\Delta y}. \qquad (6)$$

The software implementation of the considered numerical model was performed in the Fortran environment.

5. 2. Construction of a numerical model for calculating the temperature field

To predict the thermal fields inside the fairing of a launch vehicle, at the stage of pre-launch preparation, the following energy equation is used

$$\frac{\partial T}{\partial t} + \frac{\partial uT}{\partial x} + \frac{\partial vT}{\partial y} = \frac{\partial}{\partial x}\left(a_x \frac{\partial T}{\partial x}\right) + \frac{\partial}{\partial y}\left(a_y \frac{\partial T}{\partial y}\right), \qquad (7)$$

where T is the air temperature, °C; a_x, a_y are the thermal diffusivity coefficients, m^2/s; u, v are the components of the air velocity vector, m/s; (x_i, y_i) are Cartesian coordinates, m; t is time, s.

Boundary conditions for the energy equation (calculated area ABCD, Fig. 3–5) are as follows:

1) at the entrance to the calculated area: $T|_{AB} = T_{entrance}$, where $T_{entrance}$ is the known air temperature forced into the fairing (boundary AB);

2) at the flow exit boundary from the calculated area (zone D and zone C, Fig. 3–5): $\partial T/\partial n = 0$, where $\vec{n}$ is the unit normal vector to the boundary;

3) on solid surfaces: $\partial T/\partial n = 0$, where $\vec{n}$ is the unit normal vector to the boundary;

4) on the sides of the fairing: $T = \text{const}$.

Initial condition: $T_{t=0} = T_0$, where T_0 is the known temperature for time $t = 0$.

To build this numerical model, the following transformations are performed for the derivatives from equation (7):

$$\frac{\partial uT}{\partial x} = \frac{\partial u^+ T}{\partial x} + \frac{\partial u^- T}{\partial x}, \qquad (8)$$

$$\frac{\partial vT}{\partial y} = \frac{\partial v^+ T}{\partial y} + \frac{\partial v^- T}{\partial y}, \qquad (9)$$

$$u^+ = \frac{u + |u|}{2}, \quad u^- = \frac{u - |u|}{2}, \qquad (10)$$

$$v^+ = \frac{v + |v|}{2}, \quad v^- = \frac{v - |v|}{2}. \qquad (11)$$

Next, the following approximation of derivatives is performed:

$$\frac{\partial u^+ T}{\partial x} \approx \frac{u_{i+1,j}^+ T_{i,j}^{n+1} - u_{i,j}^+ T_{i-1,j}^{n+1}}{\Delta x} = T_x^+ C^{n+1}, \qquad (12)$$

$$\frac{\partial u^- T}{\partial x} \approx \frac{u_{i+1,j}^- T_{i+1,j}^{n+1} - u_{i,j}^- T_{i,j}^{n+1}}{\Delta x} = L_x^- T^{n+1}, \qquad (13)$$

$$\frac{\partial v^+ T}{\partial y} \approx \frac{v_{i,j+1}^+ T_{i,j}^{n+1} - v_{i,j}^+ T_{i,j-1}^{n+1}}{\Delta y} = L_y^+ T^{n+1}, \qquad (14)$$

$$\frac{\partial v^- T}{\partial y} \approx \frac{v_{i,j+1}^- T_{i,j+1}^{n+1} - v_{i,j}^- T_{i,j}^{n+1}}{\Delta y} = L_y^- T^{n+1}, \qquad (15)$$

$$\frac{\partial}{\partial x}\left(\mu_x \frac{\partial T}{\partial x}\right) \approx \mu_x \frac{T_{i+1,j}^{n+1} - T_{i,j}^{n+1}}{\Delta x^2} - \mu_x \frac{T_{i,j}^{n+1} - T_{i-1,j}^{n+1}}{\Delta x^2} = M_{xx}^- T^{n+1} + M_{xx}^+ T^{n+1}, \qquad (16)$$

$$\frac{\partial}{\partial y}\left(\mu_y \frac{\partial T}{\partial y}\right) \approx \mu_y \frac{T_{i,j+1}^{n+1} - T_{i,j}^{n+1}}{\Delta y^2} - \mu_y \frac{T_{i,j}^{n+1} - T_{i,j-1}^{n+1}}{\Delta y^2} = M_{yy}^- T^{n+1} + M_{yy}^+ T^{n+1}. \qquad (17)$$

The differential splitting scheme for numerical integration of equation (7) is written as:

– the first splitting step $k = n + 1/4$, $c = n$

$$\frac{T_{i,j}^k - T_{i,j}^c}{\Delta t} + L_x^+ T^k + L_y^+ T^k = 0.25\left(M_{xx}^+ T^k + M_{yy}^+ T^k + M_{xx}^- T^c + M_{yy}^- T^c\right), \qquad (18)$$

– the second splitting step $k = n + 1/2$, $c = n + 1/4$

$$\frac{T_{i,j}^k - T_{i,j}^c}{\Delta t} + L_x^- T^k + L_y^- T^k = 0.25\left(M_{xx}^+ T^c + M_{yy}^+ T^c + M_{xx}^- T^k + M_{yy}^- T^k\right), \qquad (19)$$

– in the third step of the splitting, equation (19) is solved;

– in the fourth step of the splitting, equation (18) is solved.

Thus, the solution to equation (7) is based on the cyclic solution to difference equations (18), (19).

5. 3. Conducting computational experiments based on the constructed numerical models

Below are the results of solving the problem of determining the efficiency of using the screen based on the constructed numerical models. The task was to determine the temperature near the "sensitive" sides of the payload – this is the S–E line (line 3) and the K–E line (Fig. 3–5). According to the customer's conditions, the air temperature near these surfaces should not be more than 25°C. When conducting the computational experiment, the calculated area had the following dimensions: length B–C – 4.99 m, width A-B – 4 m (Fig. 3–5). At the A–B boundary, a uniform air flow with a speed $U = 2$ m/s was set, and the fairing body temperature was 40°C. If additional air is injected into the fairing (Fig. 5), then the velocity of the injected air flow is $U = 2$ m/s, the temperature of the injected air flow is 25°C.

Fig. 6 shows the temperature field in the main fairing for the first scenario (no protective screen), when the temperature of the injected air is 25°C.

Fig. 6 demonstrates that the air temperature near the sensitive surfaces of the payload is in the range of 28–30°C. Thus, the customer's conditions are not met. Fig. 7 shows in more detail how the temperature value changes along the sensitive surfaces. Hereafter, the value $x = 0$ corresponds to point S (plot 1) and point K (plot 2).

Fig. 7 demonstrates that the highest temperature occurs near the corner points S and K; this temperature is on the order of 30–31°C. Thus, there is a violation of the customer's conditions, which requires a change in the temperature regime inside the fairing. As a first step towards solving this problem, it makes sense to investigate how the temperature regime changes when air with a lower temperature is supplied inside the fairing. Fig. 8 shows the temperature field inside the fairing if the temperature of the air being blown is 15°C (boundary A–B Fig. 3), i.e., 10°C less than in the previous version.

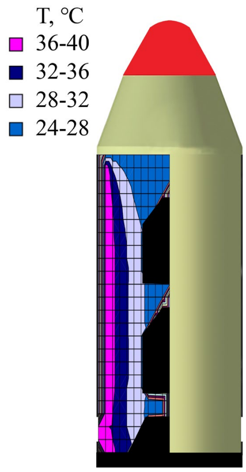

Fig. 6. Temperature field in the fairing in the absence of a screen, inlet air temperature is 25°C (scenario No. 1)

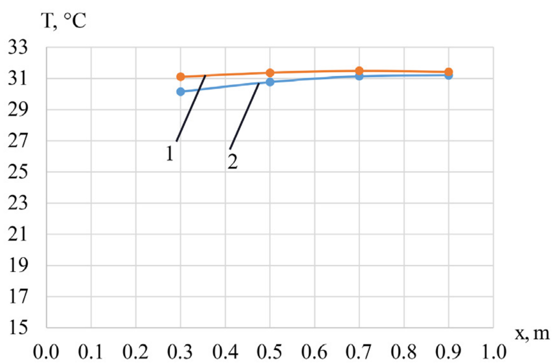

Fig. 7. Temperature near sensitive surfaces (scenario #1), inlet air temperature is 25°C: 1 – *S–E* surface; 2 – *K–E* surface

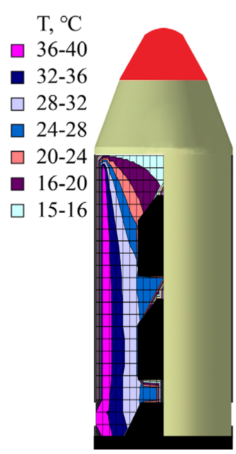

Fig. 8. Temperature field in the fairing in the absence of a screen, air temperature of the air blown in is 15°C (scenario No. 1)

Analysis of the data presented in Fig. 8 reveals that a decrease in the temperature of the air blown by 10°C did not lead to a significant change in the temperature distribution near the sensitive surfaces of the payload. This can be seen in more detail in Fig. 9, which shows the temperature distribution along the sensitive surfaces.

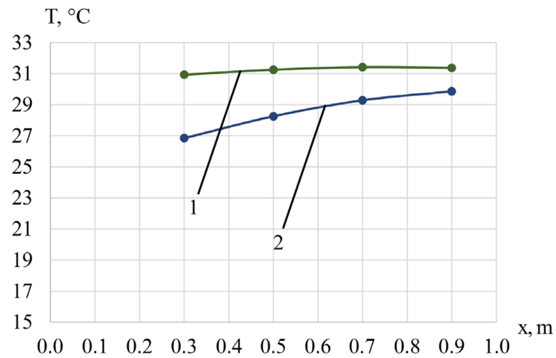

Fig. 9. Temperature near sensitive surfaces (scenario No. 1) temperature of the air blown in is 15°C: 1 – surface *S–E*; 2 – surface *K–E*

Fig. 9 demonstrates that the injection of air with a lower temperature did not affect the temperature reduction near the sensitive surfaces.

Thus, another approach is proposed to reduce the temperature near the sensitive surfaces of the payload – the use of a protective screen. The first stage of our study considers the use of a long screen (Fig. 4). Fig. 10 shows the temperature field in the main fairing for this scenario, when the temperature of the injected air is 25°C, the screen is located at a length of 20 cm from the surface of the payload.

Fig. 11 shows how the temperature value changes along the sensitive surfaces for this scenario.

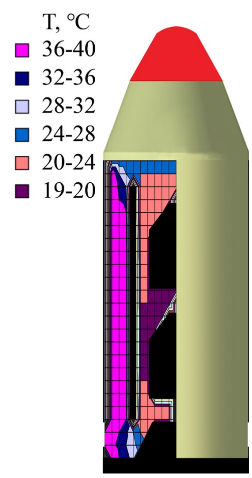

Fig. 10. Temperature field in the fairing with a long screen, inlet air temperature is 25°C (scenario No. 2)

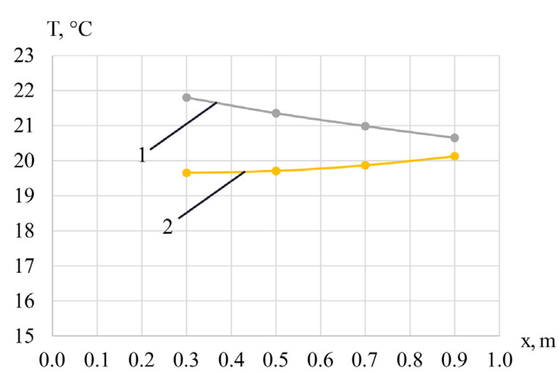

Fig. 11. Temperature near sensitive surfaces (scenario #2): 1 – surface *S–E*; 2 – surface *K–E*

Fig. 11 demonstrates that the use of a long screen makes it possible to provide the desired temperature value near the sensitive surfaces near the sensitive surfaces of the payload.

Fig. 12 shows the temperature field in the main fairing for the third scenario, when the temperature of the air blown is 25°C and two short screens are used at a length of 20 cm from the payload surface (blow at the boundary A–B, Fig. 5).

Fig. 13 shows how the temperature value changes along the sensitive surfaces for scenario No. 3.

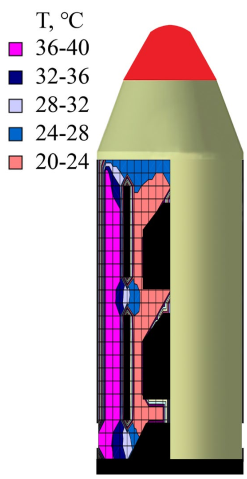

Fig. 12. Temperature field in the fairing with two short screens, inlet air temperature is 25°C (scenario No. 3)

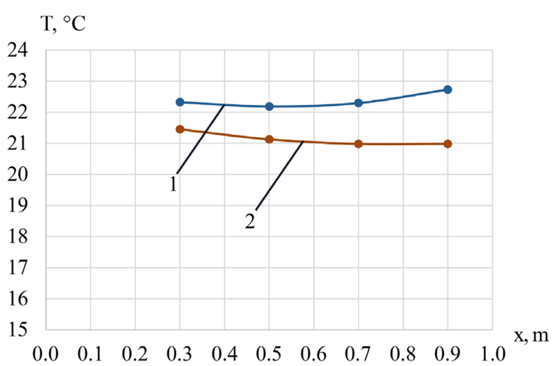

Fig. 13. Temperatures along sensitive surfaces for scenario No. 3: 1 – *S–E* surface; 2 – *K–E* surface

The results shown in Fig. 12, 13 demonstrate that the use of two short screens makes it possible to reduce the temperatures to a range that meets the customer's conditions. It should be noted that the use of two short screens makes it possible to reduce the "non-usable" mass of the protective agent (screen) inside the fairing, which is very important for the tasks of this class.

Fig. 14 shows the temperature field in the main fairing for the fourth scenario, when the temperature of the air blown is 25°C (border A–B, Fig. 5), two short screens are used at a length of 20 cm from the surface of the payload and there is an additional lateral air blow with a temperature of 25°C.

Fig. 15 shows how the temperature value changes along the sensitive surfaces for scenario #4.

Fig. 14, 15 show that the use of two short screens + blowing also makes it possible to reduce temperatures to a range that meets the customer's conditions.

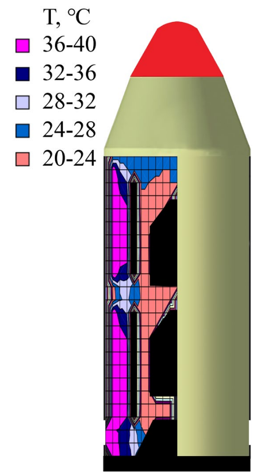

Fig. 14. Temperature field in the fairing in the presence of two short screens + blowing, blowing air temperature is 25°C (scenario #4)

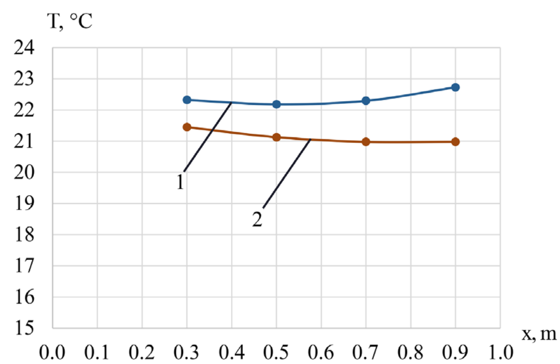

Fig. 15. Temperatures along sensitive surfaces for scenario #4: 1 – surface *S–E*; 2 – surface *K–E*

It should be noted that the calculation time of each scenario is 2 s.

6. Research on the construction of a numerical model for estimating air temperature in the main fairing of a launch vehicle: results and summary

The numerical model for solving the aerodynamic problem uses explicit difference formulas (4), (5) to determine the velocity potential field. This makes it possible to carry out its simple software implementation for calculations in areas of complex geometric shape (Fig. 3–5).

The numerical model, built for calculating the temperature field in the main fairing, is based on the split energy equation. Using this splitting makes it possible to derive explicit formulas (18), (19) to determine the temperature in the main fairing. Using such splitting, the software implementation of equations (18) and (19) is significantly simplified.

The results of our computational experiment show a significant influence of the screen on the formation of the temperature field inside the fairing. Using the screen makes it possible to reduce the temperature near the sensitive surfaces of the payload, but the pattern of temperature distribution near these surfaces is different for a long screen and two short screens. Thus, for a long screen (Fig. 10, 11), the temperature decreases along the first sensitive surface and gradually increases along the second sensitive surface, but, near both surfaces, the temperature is within the limits formulated by the customer – no more than 25°C. But when using two short screens, on the contrary, the temperature gradually increases along the first sensitive surface and decreases along the second sensitive surface (Fig. 12, 13); for this scenario, the temperature is also within the limits formulated by the customer. The same pattern occurs when using two short screens + additional air supply (Fig. 14, 15). This can be explained by the fact that the "geometric" influence (the presence of screens) on the temperature field prevails over the "mass" influence – additional air supply to the fairing. This result should be expected since the blow takes place in a significantly limited space of the fairing and in the presence of a number of obstacles in this space (screens, payload body).

It is necessary to emphasize the peculiarity of the proposed mathematical model – the speed of calculation of the temperature field on low and medium power computers. The speed of computer calculation is based on the use of the model of potential air motion for solving the aerodynamic problem. If the Navier-Stokes model is used for solving the aerodynamic problem (for example, the ANSYS program package [13, 14]), then several hours of computer time are required to determine the temperature field. The use of the proposed model makes it possible, in practice, to carry out a series of calculations in a short time, which is especially important at the "fore-sketch" stage of design work, when various options for technical solutions to reduce the temperature near the surfaces of the payload are considered.

As a limitation of our results, it should be noted that they could be used only for the CYCLONE – IV launch vehicle.

The disadvantage of the proposed mathematical model of heat transfer in the fairing is that the model does not take into account the influence of the viscosity of the air flow when solving the problem of flow around the payload. With this approach, it is impossible to determine the influence of the boundary layer on the formation of air temperature near the solid surface of the payload. It should be noted that the construction of a mathematical model that would take into account the influence of the boundary layer on the formation of temperature requires the integration of additional equations of continuum mechanics, which significantly increases the calculation time on the computer.

Further advancement of our model involves the design of an aerodynamics module that could allow us to calculate the flow velocity field based on the three-dimensional Navier-Stokes equations.

7. Conclusions

1. We have constructed a numerical model of aerodynamics inside the main fairing at the stage of its pre-launch preparation. The velocity field is determined by numerical integration of the equation for the velocity potential. The aerodynamics numerical model built makes it possible to take into account the presence of a protective screen inside the main fairing, as well as an additional, lateral air blow inside the fairing. An important feature of the constructed numerical model of aerodynamics is the ability to take into account the complex geometric shape of the payload when conducting a computational experiment, in particular, when arranging it according to the "tandem" scheme.

2. A numerical model has been constructed for the analysis of air temperature fields inside the main fairing during its thermostating, which takes into account the geometric shape of the payload, the location and intensity of the air blow inside the fairing, the location and dimensions of the protective screen.

3. Our computational experiments have shown that the most rational engineering solution to reduce temperature near the sensitive surfaces of a payload is to use two short screens.

Conflicts of interest

The authors declare that they have no conflicts of interest in relation to the current study, including financial, personal, authorship, or any other, that could affect the study, as well as the results reported in this paper.

Funding

The study was conducted without financial support.

Data availability

All data are available, either in numerical or graphical form, in the main text of the manuscript.

Use of artificial intelligence

The authors confirm that they did not use artificial intelligence technologies when creating the current work.

Authors' contributions

Mykola Biliaiev: Conceptualization, Project administration; **Viktoriia Biliaieva**: Formal analysis; **Tetiana Rusakova**: Formal analysis; **Vitalii Kozachyna**: Methodology software; **Pavlo Semenenko**: Methodology software; **Oleksandr Berlov**: Data curation; **Pavlo Kirichenko**: Software; **Nataliia Hrudkina**: Writing-original draft; **Yuliia Voitenko**: Resources; **Olena Dolzhenkova**: Resources.

References

1. Groves, C. E. (2014). Dissertation defense computational fluid dynamics uncertainty analysis for payload fairing spacecraft environmental control systems. Florida, 160. Available at: https://ntrs.nasa.gov/api/citations/20140008549/downloads/20140008549.pdf
2. Alcayde, V., Vercher-Martínez, A., Fuenmayor, F. J. (2021). Thermal control of a spacecraft: Backward-implicit scheme programming and coating materials analysis. Advances in Space Research, 68 (4), 1975–1988. https://doi.org/10.1016/j.asr.2021.03.041
3. New Glenn Payload User's Guide. Revision C. NGPM – MA0001 (2018). Blue Origin. Available at: https://yellowdragonblog.com/wp-content/uploads/2019/01/new_glenn_payload_users_guide_rev_c.pdf

4. Potapov, A. M., Kovalenko, V. A., Kondratiev, A. V. (2015). Payload fairings comparison of existing and prospective launch vehicles and their foreign counterparts. Aviacinno-kosmicheskaya tehnika i tehnologiya, 1 (118), 35–43. Available at: https://www.researchgate.net/publication/295920078_Sravnenie_golovnyh_obtekatelej_susestvuusih_i_perspektivnyh_otecestvennyh_raket-nositelej_i_ih_zarubeznyh_analogov
5. Kashanov, A. E., Degtyarev, A. V., Gladkiy, E. G., Baranov, E. Yu. (2012). Ocenka tehnicheskih riskov pri puske rakety-nositelya "Dnepr". Aviacionno-kosmicheskayatehnika i tehnologiya, 5 (92), 113–117.
6. Timoshenko, V. I., Agarkov, A. V., Moshnenko, Yu. I., Sirenko, V. N., Knyshenko, Yu. V., Lyashenko, Yu. G. (1999). Problemy termostatirovaniya i obespecheniya sohrannosti kosmicheskogo apparata v period predstartovoy podgotovki i pri vyvedenii na orbitu. Kosmichna nauka i tekhnolohiya, 5 (5/6), 56–64. Available at: https://www.mao.kiev.ua/biblio/jscans/knit/1999-05/knit-1999-05-5-6-09-timoshenko.pdf
7. Falcon User's Guide. Version 8 (2025). SPACEX. Available at: https://www.spacex.com/assets/media/falcon-users-guide-2025-05-09.pdf
8. LM-3A Series Launch Vehicle User's Manual (2011). Available at: http://www.georing.biz/usefull/LM-3A%20Series%20Launch%20Vehicles%20User's%20Manual%20Issue%202011.pdf
9. Ariane 6 User's Manual. Issue 2 Revision 0. Arianespace. Available at: https://ariane.group/app/uploads/sites/4/2024/10/Mua-6_Issue-2_Revision-0_March-2021.pdf
10. Pamuk, M. T. (2019). Numerical simulation of forced convection in an enclosure. JP Journal of Heat and Mass Transfer, 18 (1), 133–144. https://doi.org/10.17654/hm018010133
11. Dutra, M., Monteiro, M., Martí, A. C. (2024). Cooling of an object by forced convection. European Journal of Physics, 45 (4), 045101. https://doi.org/10.1088/1361-6404/ad5393
12. Riaz, A., Ibrahim, A., Bashir, M. S., Abdullah, M., Shah, A., Quddus, A. (2022). Convection Heat Transfer from Heated Thin Cylinders Inside a Ventilated Enclosure. Semiconductor Science and Information Devices, 4 (2), 10–16. https://doi.org/10.30564/ssid.v4i2.4719
13. Lou, Y.-Y., Cai, B.-Y., Li, Y.-Z., Li, J.-X., Li, E.-H. (2020). Numerical Simulation of the Air Cooling System for Scientific Payload Rack on a Space Station. Energies, 13 (22), 6145. https://doi.org/10.3390/en13226145
14. Junxin, Z., Wenan, Z., Zhongye, W., Laping, X., Gang, L., Chuang, Z. et al. (2021). Strategy for Enhancing Fairing Air Conditioning Security Capacity in the On-Time Launch of Chinese Mars-1 Exploration. Journal of Physics: Conference Series, 2006 (1), 012060. https://doi.org/10.1088/1742-6596/2006/1/012060
15. Biliaiev, M., Rusakova, T., Biliaieva, V., Kozachyna, V., Berlov, O., Semenenko, P. (2022). Analysis of Temperature Field in the Transport Compartment of the Launch Vehicle. 26th International Scientific Conference Transport Means 2022, 122–127. Available at: https://transportmeans.ktu.edu/wp-content/uploads/sites/307/2023/02/Transport-Means-2022-Part-I.pdf