

MSC2010: 76D05, 80A20, 65N06

© *A. A. Fomin, L. N. Fomina*

## NUMERICAL SOLUTION OF THE HEAT TRANSFER PROBLEM IN A SHORT CHANNEL WITH BACKWARD-FACING STEP

A test problem of the laminar steady incompressible flow and heat transfer over backward-facing step in a 2D short channel is presented. The focus of the study is on the changes in heat transfer characteristics of the flow field inside the channel due to different boundary conditions for heat flux at the outflow border of the domain. The Navier–Stokes equations in a velocity-pressure formulation and energy equation are numerically solved using a uniform grid of  $6001 \times 301$  points. The control-volume technique for the second-order difference approximation for spatial derivatives is used. The solutions were validated for a wide range of Reynolds numbers ( $100 \leq Re \leq 1000$ ) and Prandtl number  $Pr = 0.71$ , comparing them to experimental and numerical results found in the literature. The isotherm patterns and behaviors of Nusselt number along the heated bottom wall of the channel are examined. The study results showed that a condition for the heat flow (temperature) at the outlet border can influence the heat transfer in the whole domain. The nonlinear boundary condition for temperature at the outflow border is claimed as the best.

*Keywords:* incompressible fluid flow, heat transfer, outflow boundary condition.

DOI: [10.20537/vm170311](https://doi.org/10.20537/vm170311)

### Introduction

#### *Aim and objective of the work*

Heat transfer and fluid flow in separated streams are found in heat exchangers, combustors, microelectronic circuit boards, and so on. Hence, the investigation of separated flows is a problem of great importance for fundamental and industrial reasons. As is well known, the flow over a backward-facing step is the most representative issue of the class of separated flows since it has a configuration which is regarded as having the simplest geometry. In other words, the heat and mass transfer in a 2D plane channel with a backward-facing step and a heated bottom wall (or any other one) is considered in numerous experimental and numerical investigations. Thus, it is a classic fluid mechanics benchmark problem.

Because heat transfer and fluid flow characteristics experience large variation within separated regions, it is very essential to carry out an accurate simulation of the mechanisms of heat transfer in such regions in order to enhance or reduce heat flows. It is clear that a solution of the problem first of all depends on the Reynolds number  $Re$ , the Prandtl number  $Pr$ , and the expansion ratio of the channel  $ER$  which is defined as the ratio of the height of outflow border of the channel to the height of its inlet segment. Boundary conditions are also of no less importance in the mathematical formulation of the problem [1]. The fluid flow and heat transfer characteristics at the inlet border and on the walls of the channel, as a rule, are known. So, it is not difficult to define boundary conditions at these boundaries of the domain. An absolutely different situation takes place at the outlet boundary of the channel where the field characteristics of heat and mass transfer are unknown and need to be computed.

In general, formulation of a condition at the so called ‘open boundary’ (for the present problem it is the outflow border) isn’t a fully resolved issue to the present day. The main feature of an ideal open boundary condition is to allow a flux of any characteristic of the stream to exit the domain through the open boundary without any upstream perturbances [2]. As to the present study, it means that the heat waves must cross outflow border without any blocking or conversely fast escape. As a rule,

it is expressed in a sharp increase or reduction of fluid flow temperature within a zone near the outlet border of the channel.

Numerous investigations are devoted to studying incompressible fluid flows in domains with open boundaries (see, for example, [3–8]). But in most cases it concerns the conditions for dynamic characteristics of the flow. In regard to a transport unknown  $\Phi$  (i.e., temperature, mass concentration and so on), there are only a few studies that investigate open boundary conditions for those quantities. Although, as will be shown later, open boundary conditions for heat flux (temperature) can substantially influence the field of temperature in the whole domain.

Thus, the objective of this study is a numerical investigation of the influence of open boundary conditions for heat flux (temperature) on a solution of the problem of the 2D backward-facing step flow of incompressible heat-conductive fluid.

### *Literature Survey*

The fluid flow and heat transfer characteristics downstream of the backward-facing step in a plane channel have been studied by many researchers. As usual, experimental studies are presented by a relatively small amount of works [9–12]. Parameters of heat and mass transfer in these studies are measured directly. Therefore, there is no problem of influence of boundary conditions on the research results. At the same time practically in all theoretical studies the domain length is accepted so that at the exit from a channel the fluid flow is, at least, a plain-parallel stream. It is considered that in this case the reverse influence of an open boundary condition is negligible for transport quantity  $\Phi$ . As a rule, a fee for the similar approach is a needless long size of the calculation domain.

Characteristics of heat transfer at the reattachment region downward a backward-facing step were investigated experimentally by Kawamura et al. [9]. They reported that instantaneous heat transfer distribution changed in a complex manner and, therefore, a temporal and spatial universal distribution of a heat transfer coefficient was not found. But in average the peak value of the heat transfer distribution shows a maximum at about one step height upstream from the time-averaged reattachment point of the fluid flow. The same problem, namely, the location of Nusselt number maximum relative to the point of reattachment of the fluid flow, was investigated by Sparrow et al. [10]. The results, both experimental and numerical, were obtained for Reynolds numbers  $Re = 100, 200$  and  $300$ , and for a Prandtl number of  $0.7$ . For most of the considered cases, the point of heat transfer maximum was situated upstream of the reattachment point. This result confirmed conclusions of the paper [9] about non-equivalence of these two points of the flow.

The spatio-temporal characteristic of heat transfer accompanied by the fluid flow separation and reattachment was investigated experimentally by Nakamura et al. [11, 12]. In these studies, the heat transfer in the fluid flow reattaching region has a spot-like feature, which spreads with time and overlaps with other ones to form a complex structure. They observed that the mean Nusselt number distribution behind the reattaching region was approximately proportional to  $2/3$  power of Reynolds number. Namely, behind the reattachment point the relation of local Nusselt number to local Reynolds number in  $2/3$  degree can be approximated by a constant value of  $0.13$ . They also reported that the time-space distribution of the heat transfer has a typical spanwise wavelength and a typical fluctuating frequency in the reattaching region. Furthermore, the origin of the spanwise periodicity is not the instability upstream of the flow separation, but is due to some instability, which is accompanied by the flow separation and reattachment.

Aung et al. [13] have presented theoretical results concerning the flow and heat transfer in laminar flow past a backward-facing step. They reported that the size of the initial boundary layer in the inlet part of the domain before step can have opposing effects on the reattachment distance, depending on whether the Reynolds number is held constant. Also, it was shown that computed velocity and temperature profiles had major discrepancies in the separated region after step with those given by the simple theory.

Kondoh [14] studied laminar heat transfer in a separating and reattaching flow downstream a backward-facing step. Three parameters governing the heat transfer:  $Re$ ,  $ER$ ,  $Pr$  were varied in this research; Reynolds number from  $10$  to  $500$ , Prandtl number from  $0$  to  $1000$ , and expansion

ratio from 1.25 to 2. Here it should be noted that solutions of the problem weakly differ from each other at  $ER < 2$ , while at  $ER > 2$  they change more strongly (see, for example, [15–17]). According to the results obtained, the authors have claimed that the peak of the local Nusselt number isn't necessarily located at or very near the point of the flow reattachment. Moreover, its location considerably depends upon both the Reynolds number and the channel expansion ratio.

Valencia et al. [15] studied the incompressible laminar flow and heat transfer of air in a channel with a backward-facing step for steady and pulsatile inlet conditions. The Reynolds number was varied from 100 up to 1250, and the expansion ratio from 0.25 up to 0.75. It was found that heat transfer in the bottom wall of the channel had maximum value downstream the reattachment length for steady flows. For the pulsating flow the primary vortex arose on the corner of the backward-facing step wall at the maximum inlet velocity, and filled the channel at the minimal inlet velocity. Also, the time-average pulsatile heat transfer at the channel walls was higher than with steady flow at the same mean Reynolds number.

The numerical prediction of the fluid flow and heat transfer characteristics for a backward-facing step by discharging a jet through the roof wall of the channel perpendicularly to the main flow was carried out by Yang et al. [18]. The predicted attachment point and local heat transfer coefficient were in good agreement with the experiments. A dependence of local heat transfer coefficient from the jet position has been found.

The heat transfer enhancements of backward-facing step flow in a channel through the solid or slotted baffle installation on the roof channel wall have been studied by Tsay et al. [19]. The main parameters of the investigation were  $Pr = 0.7$ ,  $ER = 0.5$ ,  $H_b = 0.3$ ,  $W_b = 0.2$ ,  $50 \leq Re \leq 500$ , and  $0 \leq Gr/Re^2 \leq 1$ . Here  $H_b$  is the baffle height,  $W_b$  is the baffle thickness, and  $Gr$  is the Grashof number. The authors reported that a slight movement of the baffle could cause a drastic change in the fluid flow structure and temperature field in the channel. In addition, they stated that the effects of baffle width on heat transfer were insignificant. Also, in these articles it was shown that a slotted baffle can enhance the average Nusselt number for the heating section of the channel wall by the maximum of 190 %. As for the solid baffle, the enhancement may be up by 230 %.

A study on mixed convective heat transfer for two-dimensional laminar flow in an inclined channel with a backward-facing step was presented in [20]. The uniform heat flux came into the channel through a wall behind a step. At the same time, the opposite wall of the channel was cold. The inlet flow was fully developed and was at a uniform temperature. The investigation was carried out at  $Re = 100$ , Grashof number  $Gr = 609$ , and expansion ratio  $ER = 2$ . The inclination angle  $\gamma$  was varied from 0 to  $360^\circ$  at  $Pr = 0.712$ , and Prandtl number  $Pr$  – from 0.07 to 100 at  $\gamma = 0^\circ$ . As was shown, increasing  $\gamma$  from 0 to  $180^\circ$  increased the reattachment length, but it decreased the wall friction coefficient and  $Nu$  at the heated wall. Also, increasing  $Pr$  increased  $Nu$  and the reattachment length, but it decreased the wall friction coefficient.

Forced convection heat transfer due to fluid flow over a backward-facing step with a porous floor (bottom wall of the channel) segment for a wide range of the pressure loss coefficient was investigated numerically by Abu-Hijleh [21]. The resulting Reynolds number based on the average incoming stream velocity and step height was 100, the expansion ratio  $ER$  was 2. The porous segment was positioned around the flow reattachment point on the floor. The main conclusions of the study were that the addition of a porous segment resulted in an increase in the maximum Nusselt number  $Nu$ , and maximum  $Nu$  was more affected by changes in the axial rather than the transverse pressure loss coefficient of the porous floor segment.

Batenko et al. [22] carried out a numerical study of a laminar separated flow behind a rectangular step on a porous bottom wall of a plane duct with uniform gas injection or suction through the porous surface. The problem was solved for the following parameters:  $10 \leq Re \leq 1000$ ,  $ER = 2$ ,  $10^{-4} \leq |F| \leq 10^{-1}$ , where  $F$  is the ratio of the velocity of gas injection (suction) to the mean-mass velocity at the cross inlet section of the duct. It was shown that mass transfer on the porous surface caused strong changes in the flow structure and substantially affected the position of the reattachment point, as well as friction and heat transfer.

Abu-Nada et al. [23] studied a heat transfer and fluid flow over a backward-facing step under the

effect of suction and injection of warm fluid, which is implemented on the bottom wall of the channel after the step. The investigation was carried out for the following values of problem parameters:  $Pr = 0.71$ ,  $200 \leq Re \leq 800$ ,  $3/2 \leq ER \leq 3$ ,  $-0.005 \leq F \leq 0.005$ . It was shown that on the bottom wall, and inside the primary recirculation zone, suction increased the coefficient of friction and injection reduced it. The reattachment length of the primary recirculation zone was increased by increasing the injection bleed coefficient  $F$  and was decreased by increasing the suction bleed rate. Also, the local Nusselt number on the bottom wall increased by suction and decreased by injection, and the opposite occurred at the roof wall.

A steady-state conjugate heat transfer study of a wall and a fluid for a two-dimensional laminar incompressible stream over a backward-facing step was carried out by Kanna et al. [16]. As main results they demonstrated that the conjugate interface temperature value decreased along the step length and height. The minimum Nu fell after the reattachment point position. At the same time, the interface temperature value decreased when Re was increased and it decreased for higher Pr. Also, the local Nusselt number had a peak value near the inlet cross section and the second peak occurred after the reattachment point of the stream.

A somewhat unusual investigation was carried out by Saha et al. [24] in which a numerical analysis of mixed convection was observed in a rectangular enclosure with different placement configurations of the inlet and outlet openings. At the outlet of the computational domain a so-called 'convective boundary condition' was used. The authors reported that the average Nusselt number and the surface temperature on the heat source strongly depended on the positioning of the inlet and outlet fluid flow.

An investigation of backward-facing step incompressible fluid flow, heat transfer and conjugated heat transfer is presented in Teruel et al. [25]. Calculations were executed for the following parameters of the problem:  $Re = 10^{-4}, 10^{-2}, 1, 500, 800$ ;  $Pr = 0.7$ ;  $ER = 1.5$ . Excellent agreement for fluid flow at  $Re = 800$  was found with numerical data reported in the literature. However, the results obtained for heat transfer and conjugate heat transfer have shown a few differences with available data. As a result, further analysis was recommended in this paper to find the origin of this disagreement. Here at once it should be noted that the test results of the present research are in good agreement with data [25].

Gada et al. [26] have investigated different thermal boundary conditions for the fluid flow problem in a plane channel without and with phase changes. The studies in a dimensional form were carried out both by analytical and by numerical methods. Numerical 2D simulations were done for the operating conditions proposed in the analytical part of the research. An excellent agreement was found between the numerical and analytical results, with and without phase changes.

Mitsoulis et al. [27] have tested the 'free' (the same as 'open') boundary condition in several benchmark problems of viscous flow. This condition was proposed by Papanastasiou et al. [28] to handle truncated domains with so-called 'synthetic' (artificial) boundaries, where the outflow conditions were unknown. They reported that the condition was primarily suitable for flows in cases where the boundary conditions were not known and the flow phenomena downstream were determined by the conditions upstream (convective flows). At the same time, it was not suitable for the problems of flow with gravity or surface-tension effects.

The investigation by Dimakopoulos et al. [29] also concerned the use of Papanastasiou free boundary condition at the synthetic borders. But contrary to the previous research [27] the boundary condition was used not only at the outflow border of a domain, but also at inflow one. The authors stated that the open boundary condition was a very attractive alternative for imposing inflow boundary conditions in all cases of fluid flow.

As mentioned above, a quantity of types of boundary conditions at open borders for a heat flux (temperature) is less than a quantity of similar types of the boundary conditions for dynamic parameters of a fluid flow. Moreover, there are only a few works where the influence of different kinds of open boundary conditions on a problem solution was studied. In the present reference list only articles of Sani et al. [5] and Papanastasiou et al. [28] deal with this subject.

In [28] a backward-facing step flow in a plane duct problem was solved using two kinds of outflow

conditions: zero flux of an unknown and free boundary condition. It was shown that the second condition was useful not only at a far distance from the zone with eddies but it worked even when the flow was not developed at the synthetic outflow border.

Results of the solution of four fluid flow problems by using five types of open boundary conditions for the normal momentum equation at outflow border, which were obtained by nine research groups, are discussed in [5]. The main conclusion of this complex report is that the problem of implementation of an open boundary condition is still far from final solution. The authors consider that the most important issue for incompressible flows is that the incompressibility constraint is all-pervasive and reaches up both open boundaries and all other ones. As a result, the instantaneous interaction of the pressure and the normal component of the velocity define the structure of incompressible fluid flow.

In this situation, the problem of an open boundary condition for non-dynamic transport unknown  $\Phi$ , as a rule, is not so dramatic. Here the standard recommendation concerning the location of the open border is to position it as far as possible from a zone of eddy perturbations of stream [30]. In this position the open boundary condition should have no significant effect on the solution inside the calculation domain. That is why a range of variants of boundary conditions for  $\Phi$  is not so wide as it is for dynamic unknowns because the kind of an open boundary condition for  $\Phi$  isn't of great importance. Moreover, the zero first derivative of  $\Phi$  along the normal direction to the open border is the most usable condition. For example, the zero second derivative of temperature is used as an open boundary condition in [14,20,22,24], and in [27,29] these conditions were obtained from the truncated form of an energy equation. In all the other articles of the present reference list, where the energy equation is solved, the zero first derivative of temperature is used as the open boundary condition at the outflow border.

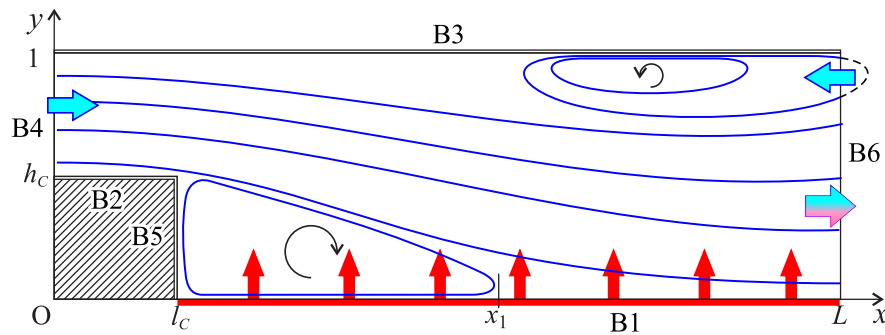
This paper is structured as follows. The physical assumptions, geometry of the domain, governing equations, and boundary conditions for viscous heat-conductive incompressible backward-facing step flow problem and definitions of main characteristic parameters of the stream are described in Section 1. In Section 2, the details of the numerical solution technique are given. Then the results of code validation in terms of mesh refinement and comparisons of present solutions with numerical ones found in the literature are presented in Section 3. Next, Section 4 contains the main results of the investigation such as the fields of streamlines and temperature as well as the profiles of Nusselt numbers which are demonstrated and discussed. Finally, in Section 5 conclusions are drawn.

## § 1. Mathematical formulation

### § 1.1. Physical assumptions and geometry of the domain of the flow and heat transfer

An incompressible heat-conductive Newtonian fluid with constant fluid properties such as density  $\rho$ , viscosity  $\mu$ , and thermal conductivity  $\lambda$  is assumed. The flow in the problem is two-dimensional, laminar and stationary. The basic scheme of the flow and heat transfer in Cartesian coordinates is shown in Figure 1. Here all distances are dimensionless. The height of the outlet section of the channel is chosen as the characteristic length. The notations of the length and the height of the backward-facing step are  $l_c$  and  $h_c$ , respectively. According to its definition, the expansion ratio  $ER$  is calculated by the formula  $ER = 1/(1 - h_c)$  in the present investigation. Designations of the domain borders B1, B2, ..., B6 are accepted in the same way as in Roache [1].

A fully developed stream with constant temperature  $T_0$  flows into the inlet section of the channel through the left border of the domain and flows out through its right border. The bottom wall is heated uniformly with constant temperature  $T_w$ . Hereafter the subscript 'w' denotes the bottom wall B1. The walls B2, B3, and B5 are heat insulated surfaces. In addition to a primary recirculation zone behind the step whose length is  $x_1$ , a secondary recirculation zone exists at the upper wall of the channel. It is assumed that the flow at the inlet border of the channel is fully developed, and conversely, because a short channel takes place in the problem, the stream in the outlet section of the channel can't be considered as the fully developed flow in the general case. In other words,



**Fig 1.** Geometry of domain and scheme of the flow and heat transfer

the stream is not only outlet flow but, in general, can have inlet part of flow at the border B6 (see Figure 1).

### § 1.2. Governing equations

The non-dimensional, non-conservative form of the continuity, momentum, and energy equations in 2D Cartesian coordinates is as follows:

$$\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} = -\frac{\partial p}{\partial x} + \frac{1}{\text{Re}} \left( \frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} \right), \quad (1)$$

$$\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} = -\frac{\partial p}{\partial y} + \frac{1}{\text{Re}} \left( \frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} \right), \quad (2)$$

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0, \quad (3)$$

$$u \frac{\partial \theta}{\partial x} + v \frac{\partial \theta}{\partial y} = \frac{1}{\text{RePr}} \left( \frac{\partial^2 \theta}{\partial x^2} + \frac{\partial^2 \theta}{\partial y^2} \right), \quad (4)$$

where  $u$  and  $v$  are the horizontal and vertical components of the velocity, respectively,  $p$  is the pressure,  $\theta$  is the relative difference of temperature defined as  $\theta = (T - T_0)/(T_w - T_0)$ .

The Reynolds number is defined as  $\text{Re} = \rho U H / \mu$  and Prandtl number is defined as  $\text{Pr} = \mu C_p / \lambda$ . Here  $H$  is the full height of the channel,  $U$  denotes the average velocity of the incoming flow, which corresponds in the laminar case to two-thirds of the maximum inlet velocity,  $C_p$  is specific heat at constant pressure. Time and pressure were nondimensionalized with  $H/U$  and  $\rho U^2$ , respectively.

The time-dependent form of the momentum equations is caused by using a relaxation technique for reaching the stationary solution of the dynamic part of the problem (equations (1)–(3)) by time marching. It is clear, that when the field of velocity is defined, arbitrary quantity of stationary heat transfer problems can be solved on the basis of this field. In particular, various solutions obtained under different open boundary conditions for the thermal flux (temperature) can be analyzed.

As is well known, the main characteristic of heat transfer at a surface within a fluid flow is Nusselt number that is the ratio of convective heat exchange to conductive one along the normal direction to a surface. For the bottom wall in the problem this number is defined as follows [22]:

$$\text{Nu} = \frac{\alpha H}{\lambda} = - \left( \frac{\partial \theta}{\partial y} \right)_w, \quad (5)$$

where  $\alpha$  is a local heat transfer coefficient in the formula for the heat flux  $q$  between heated surface and fluid stream  $q = \alpha(T_w - T_0)$ . Because  $q$  is a function of the  $x$ -coordinate, Nu in equation (5) is a function of  $x$  too.

**§ 1.3. Initial and boundary conditions**

Initial conditions for components of velocity are  $u = v = 0$ . Further in the range of time  $0 < t < t_m$  a smooth increase in stream intensity through the left border of the channel up to its maximum takes place. For  $t > t_m$  all boundary conditions become stationary allowing one to seek the solution of the problem using a relaxation method. Also for the inlet boundary B4 time-dependent fully developed velocity field is specified as a parallel flow with a parabolic horizontal component and a zero vertical component of velocity given by

$$u = f(t) \frac{6(y - h_c)(1 - y)}{(H - h_c)^2}, \tag{6}$$

$$v = 0; \tag{7}$$

where

$$f(t) = \begin{cases} 0.5 \{ \sin [0.5\pi (2t/t_m - 1)] + 1 \}, & 0 \leq t \leq t_m; \\ 1, & t_m < t. \end{cases}$$

In all calculations of the current research  $t_m$  is equal 1. It is not difficult to understand that the boundary condition (6) at  $t > t_m$  sets at the inlet border B4 the value of average velocity which is equal to 1, mass flow rate  $Q = 1 - h_c$ , and maximum incoming velocity  $u_{max} = 1.5$ .

No-slip and impermeability boundary conditions for components of velocity are used at all walls. As mentioned above, in general it is assumed that flow isn't fully developed at the outlet border B6. In this sense any open boundary conditions have to be considered as rough ones. So, in the present investigation Neumann boundary conditions owing to their simplicity and usability are used for both velocity components at this border:

$$\frac{\partial u}{\partial x} = \frac{\partial v}{\partial x} = 0. \tag{8}$$

Boundary conditions for  $\theta$  are specified as follows: at the inlet border B4  $\theta = 0$ , at the heated bottom wall  $\theta = 1$ , at the other walls  $\partial\theta/\partial n = 0$  where  $n$  is a normal direction to the wall. According to the objective of the present research declared above, several kinds of open boundary conditions at the outlet border B6 are formulated here:

– the assumption of constancy of a conductive heat flux

$$\frac{\partial^2 \theta}{\partial x^2} = 0, \tag{9}$$

this is the linear condition;

– the assumption of constancy of a full heat flux [1]

$$\frac{\partial}{\partial x} \left( u\theta - \frac{1}{\text{RePr}} \frac{\partial \theta}{\partial x} \right) = 0, \tag{10}$$

this is the linear condition;

– the assumption of constancy of a relative difference of temperature

$$\frac{\partial \theta}{\partial x} = 0, \tag{11}$$

this is the linear condition;

– and the assumption of constancy of a flux of a reciprocal relative difference of temperature

$$\frac{\partial^2 (1/\theta)}{\partial x^2} = 0, \tag{12}$$

unlike the above conditions, this one is the nonlinear condition.

## § 2. Numerical technique

Since there is no direct coupling between the continuity equation (1) and momentum equations (2)–(3) for incompressible flows, a well-known approach is to derive the Poisson equation for pressure from equations (1)–(3) and to use it instead of the continuity equation (1). As a result, the technique of splitting of physical factors [31] can be applied to solving the Navier–Stokes equations (1)–(3). In particular, this technique means that the solution should converge to a steady-state one through iterations in time. In contrast to [31], two modifications of this computational technology are used herein. First, an implicit difference scheme is used for momentum equations. Second, the Poisson equation is formulated for the increment of pressure (instead of pressure, as it was made in the original work) with zero Neumann boundary conditions at all borders of the domain. The convergence of the solution to a stationary one is checked by means of the condition

$$D_V = \frac{\|u^k - u^{k-1}\|_I + \|v^k - v^{k-1}\|_I}{\tau \|\mathbf{V}^k\|_I} < \epsilon.$$

Here  $\tau$  is a time step, the superscript ‘ $k$ ’ is an index of a time level,  $D_V$  is a rate of relative change of velocity vector  $\mathbf{V}^k$ . The criterion of accuracy of solution convergence is equal to  $\epsilon = 10^{-5}$ . This value of criterion provides the accuracy of the numerical solutions up to 6th decimal.

As pointed out above, the outlet boundary conditions in equation (8) are rough ones. Therefore, the profile of  $u$  component of velocity at this border can be changed a little in accordance with some additional condition. Details of the technology for the correction of the velocity components which allows one to put an open border in any place are given in [32].

For the approximation of time derivative terms in the momentum equations, the first-order accurate Euler implicit time-stepping scheme is applied. And for the spatial discretization of equations (1)–(4), the control-volume method with fifth-order power-law scheme of second-order accuracy is used [30]. In all calculations uniform grids are used. The resulting linear system of finite difference equations is solved with the use of the original so-called line-by-line recurrence method, accelerated in Krylov subspaces [33].

Since the governing equations for  $u$ - $v$ - $p$  and the energy equation are uncoupled, the  $u$ - $v$ - $p$  equations are first solved and their results are stored. The stored information (fields of velocity components) is then used to obtain various solutions of the energy equation.

## § 3. Fluid flow and heat transfer verification

### § 3.1. Grid testing

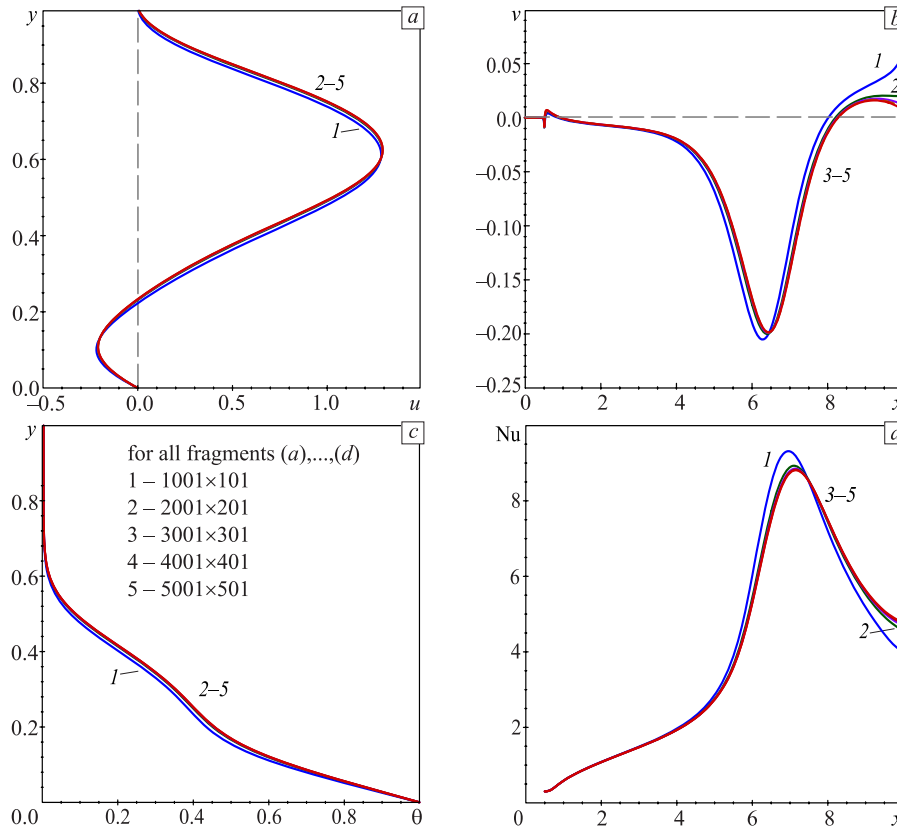
The grid refinement of solution was investigated by detailed mesh testing that was carried out for the following parameters of the problem:  $L = 10$ ,  $ER = 2$ ,  $l_c = 0.5$ ,  $Re = 1000$ , and  $Pr = 0.71$ . As is shown in Figure 2, five different meshes were examined. This testing used uniform grids of  $1001 \times 101$ ,  $2001 \times 201$ ,  $3001 \times 301$ ,  $4001 \times 401$ , and  $5001 \times 501$ .

In the figure it is clearly shown that the curves for the grid steps from  $1/300$  to  $1/500$  are almost the same with graphical accuracy. The most significant relative distinctions between solutions for the grid step of  $1/100$  and for the other grid steps reach 5–10 % for  $v$ -component of velocity and Nusselt number. For the grid step of  $1/200$  distinctions are noticeable too, but are very negligible. Thus, to increase the reliability of computing results, a grid with a step of  $1/300$  is chosen as a basis for further calculations.

### § 3.2. Comparisons with literature data

The adequacy of the present numerical technique is evaluated by comparing the present predictions of some flow and heat transfer parameters with the experimental and numerical published





**Fig 2.** Evaluation of grid independency for (a)  $u$ -component of velocity at  $x = 5.5$ , (b)  $v$ -component of velocity at  $x = 5.5$ , (c) relative difference of temperature  $\theta$  at  $y = 0.5$ , (d) Nusselt number at  $y = 0$

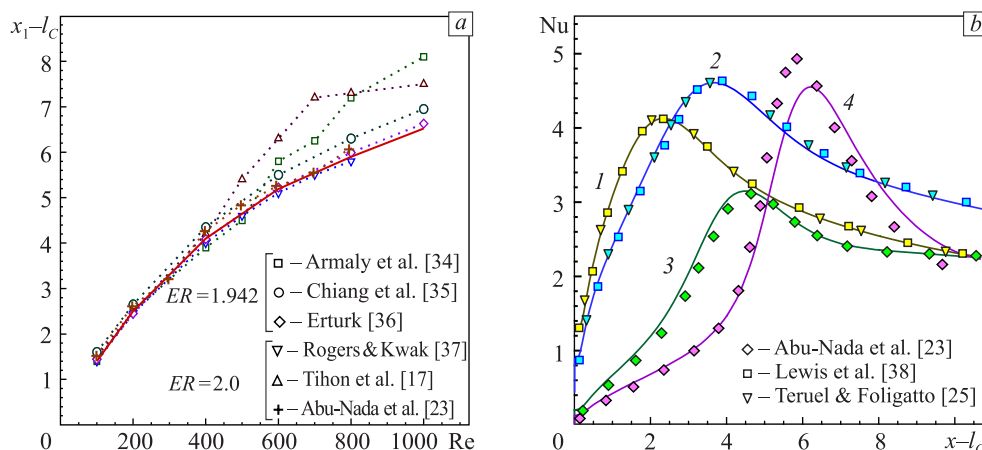
data [17, 23, 25, 34–38]. As typical results of these comparisons, Figure 3 shows the predicted and published dependencies of length of the primary recirculation zone behind the step on Reynolds number, and profiles of Nusselt number at the heated bottom wall B1.

For the range of moderate  $Re$  from 100 to 500 (Figure 3, a) a predicted length of the primary recirculation zone matches well with experimental as well as computational results. For higher  $Re$  it agrees well with computational data (especially with Erturk [36] and Rogers et al. [37]) and much less with the experimental one. The differences of the theoretical and experimental results for higher Reynolds numbers can be explained by the presence of three-dimensional effects in the experiments.

Comparisons of heat fluxes at B1 (Figure 3, b) demonstrated a good agreement of predicted  $Nu$ -profiles with those presented in the literature. Almost exact coincidence of results can be considered as a proof of the correctness of the thermal problem solution even if one takes into account a small divergence about 8–10 % at local maximum of  $Nu$  profiles at  $Re = 800$  (a curve 4). The use of quantity  $x - l_c$  instead of  $x$  in both fragments of Figure 3 indicates an insignificant dependence of the step length on the final result of the study. However, as test calculations showed, this insignificant dependence takes place for  $l_c \geq 0.5$ . But, for example, for the case  $l_c = 0$  the solution will be different from the one for the case  $l_c > 0$  (see, for example, [39]).

#### § 4. Computed results and discussion

The technique of the study consists in the following. The constant parameters of all calculations are  $Re = 1000$ ,  $Pr = 0.71$ ,  $L = 20$ ,  $l_c = 0.5$ , the resolution of a uniform difference grid is  $6001 \times 301$ . For these conditions the fields of velocity components were obtained for  $h_c = 0.5$  and  $h_c = 0.9$ . In the second case of  $h_c$  a very specific flow field takes place. A series of solutions of the heat transfer problem for different kinds of the outlet open boundary condition is calculated for each variant of the step height. The solutions are compared with each other for every series in such parameters as

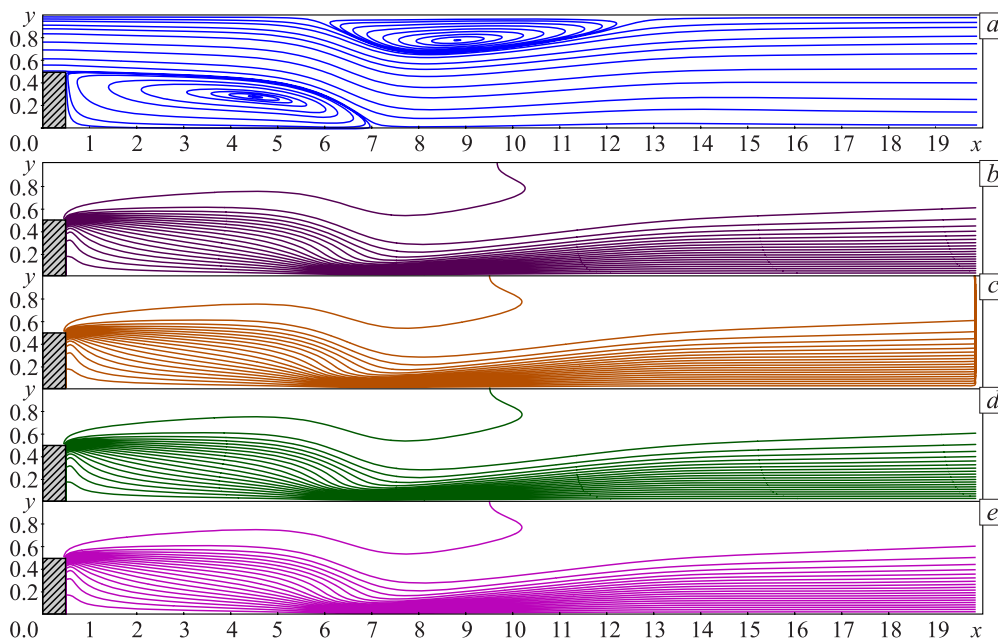


**Fig 3.** Comparison of present results (solid lines) with literature data for (a) length of the primary recirculation zone behind the step, (b) Nusselt number at B1: 1 –  $ER = 1.5$ ,  $Re=300$ ; 2 –  $ER = 1.5$ ,  $Re=600$ ; 3 –  $ER = 2.0$ ,  $Re=400$ ; 4 –  $ER = 2.0$ ,  $Re=800$

fields of isotherms and profiles of heat flux (Nusselt number) at the heated wall B1. On the basis of these comparisons the conclusions about the degree of correctness of the used open boundary conditions at the outlet border of the channel for the energy equation are drawn.

#### § 4.1. The step height $h_c = 0.5$ ( $ER = 2$ )

The fields of streamlines and isotherms for  $ER = 0.5$  are presented in Figure 4. The patterns are zoomed out along the horizontal direction.

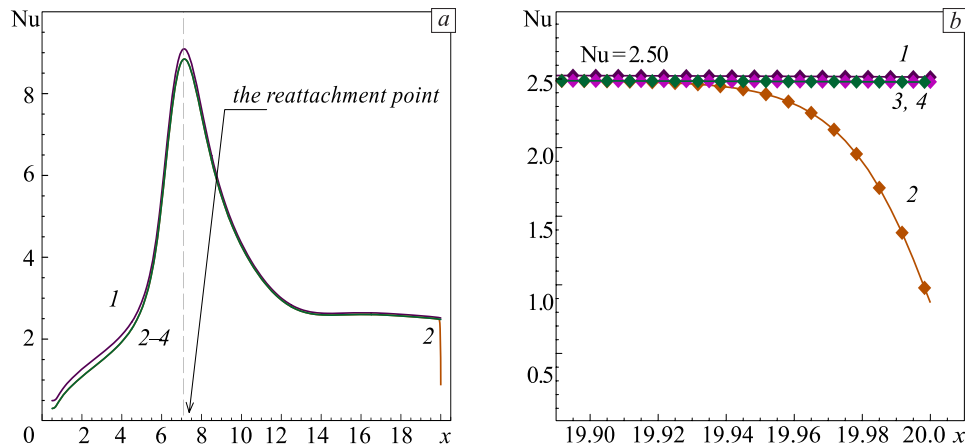


**Fig 4.** Fields of the fluid flow and heat transfer for  $ER = 2$ . (a) Pattern of the streamlines. Patterns of the isotherms for different boundary conditions at B6: (b) condition (9), (c) condition (10), (d) condition (11), (e) condition (12). Levels of  $\theta$  are: from 0.95 to 0.05 with decrement of 0.05, and 0.001

Predictably, two significant recirculation zones are situated in the flow: the first one is at the bottom wall immediately behind the step; the second one is a little further at the roof wall (Figure 4, a) compare, for example, with Erturk [36]). Fluid flow near the outlet border is unidirectional. However, its streamlines aren't absolutely parallel to each other there. Therefore, the flow is not fully developed, i.e., the cross-channel profile of the  $u$ -component of the velocity is not parabolic

and  $v$ -component is not zero. In the figure it is clearly shown that patterns of isotherms for different boundary conditions at B6 almost coincide in general. The insignificant exception arises in the case of using the boundary condition (10) (Figure 4, *c*) where a sharp curvature of isotherms in the very narrow zone at the outlet border takes place.

Additional important information about differences between solutions of the heat transfer problem depending on a kind of the open boundary condition at the outlet border can be obtained from analysis of Nusselt number profiles. In Figure 5 the profiles of Nusselt number at  $ER = 2$  are demonstrated for different open boundary conditions for the whole length of the channel behind the backward-facing step and for the narrow region near the outlet border B6.



**Fig 5.** Profiles of Nusselt number for  $ER = 2$  for different boundary conditions at B6: (a) the whole length of the domain downstream behind the step, (b) near the outflow boundary of the domain. 1 – condition (9), 2 – condition (10), 3 – condition (11), 4 – condition (12)

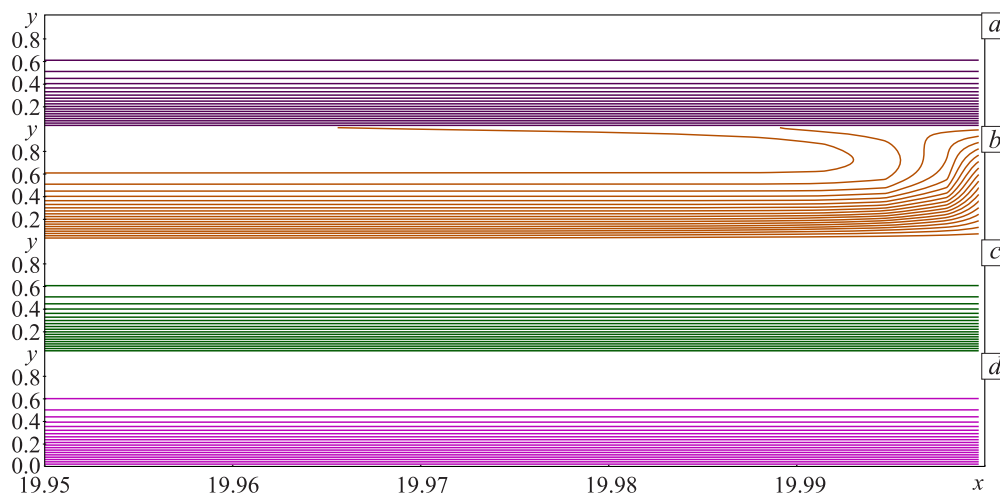
It is clearly seen that there are few differences between Nu-profiles for condition (9), on the one hand, and for conditions (10)–(12), on the other hand, in Figure 5, *a*. However, the sharp downturn of the Nu profile for condition (10) (a curve 2) near the output border is more interesting. Beyond all doubt this downturn of the curve is connected with the sharp curvatures of isotherms in Figure 4, *c*. Much more detailed behaviors of Nu-profiles close to the outlet border are presented in Figure 5, *b*. It should be noted that every second the node of the grid is marked on the curves by a ‘diamond’ bullet. The conclusion that the effect of the boundary condition (10) on the solution inside the domain covers no more than two dozen nodes of the grid follows from the behavior of curve 2 in Figure 5, *b*.

One more noticeable feature of the distributions of Nusselt number along the channel length is coincidence of positions of the Nu profiles maxima and the flow reattachment point (Figure 5, *a*). There is coincidence or almost coincidence of these two points in literature for the 2D computer modeling [14]. But in experiments the point of Nu profile maximum as a rule is localized at some distance upstream from the reattachment point of the flow [9, 10].

On the basis of Figure 5 a need arises to study the detailed structure of isotherms in the narrow zone near the outlet border for all four conditions (9)–(12). It is not difficult to see in Figure 6 that conditions (9) and (11)–(12) for the almost plane-parallel flow provide within a short distance a plane-parallel behavior of isotherms from the grid node to the node near the boundary. Whereas condition (10) leads to a sudden increase of the heat flux in vertical direction in this zone.

The reason for such behavior of the solution is obvious: condition (10) demands preservation of the full heat flux along the channel, which is the sum of convective and conductive fluxes. In other words, condition (10) doesn’t allow the heat to spread horizontally near B6 boundary. Hence, the additional heat flux which arrives from the lower more heated layers of fluid (or the wall B1) can move only up. It is clear that in this case the vertical gradient of temperature (Nusselt number) decreases.

Of course, in the issue it is possible to reduce a little the length of the calculation domain and



**Fig 6.** Patterns of the isotherms near outflow boundary of the domain for  $ER = 2$  for different boundary conditions at B6: (a) condition (9), (b) condition (10), (c) condition (11), (d) condition (12). Levels of  $\theta$  are: from 0.95 to 0.05 with decrement of 0.05

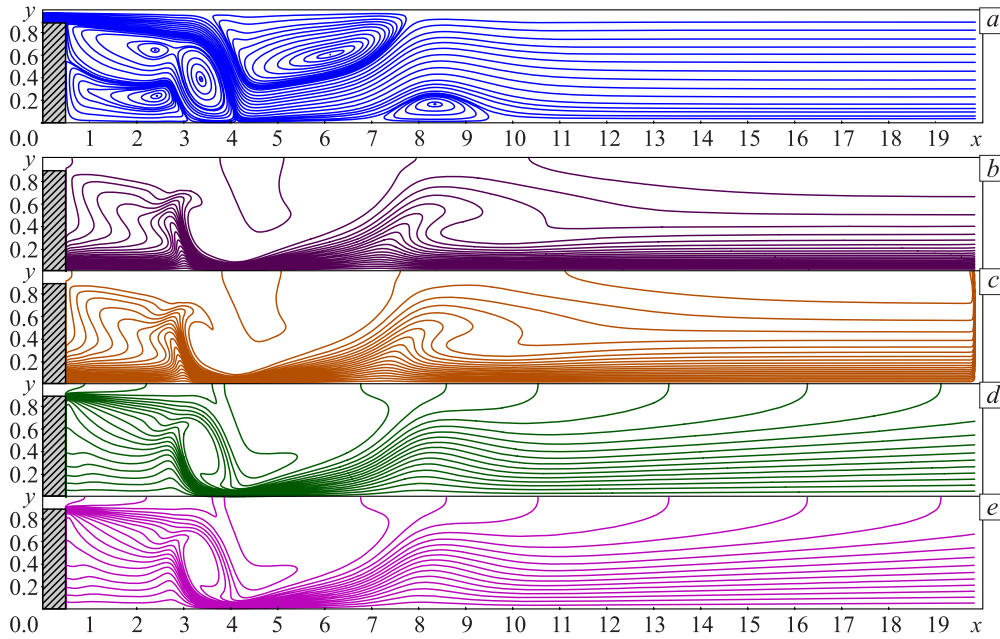
to draw the conclusion about the correctness of all the considered open boundary conditions for the variant of the problem  $ER = 2$ . In other words, in the case considered the choice of an open boundary condition for temperature has no basic value at least among widespread ones. As for the actually pattern of warming up of fluid for the channel length considered here, it is clear that a split of the stream into the top cold part of flow and the lower warm one takes place (see Figure 4). Namely, there is a localization of heat near the lower heated wall B1.

#### § 4.2. The step height $h_c = 0.9$ ( $ER = 10$ )

A fundamentally different solution takes place in the case  $ER = 10$  (see Figure 7). In this figure the patterns are zoomed out along the horizontal direction in the same way as in Figure 4. The flow has a complex circulation structure in Figure 7, a which consists of four eddies. It is interesting that there are two centers of rotation in the primary recirculation zone immediately behind the step and a saddle point between them, respectively. At the same time the structure of outlet stream is plane-parallel, i.e., it coincides with the outlet flow in the previous case (compare with Figure 4, a). As to solution of the energy problem, here another situation takes place: the patterns of isotherms for the first two variants for conditions (9)–(10) are drastically different from the other ones for the second two variants for conditions (11)–(12).

Here it should be kept in mind that all four patterns of the isotherms are correct solutions of the mathematical formulations of the problem which differ only by output boundary conditions. But the physical correctness of solutions in Figures 7, b–c raises doubts for the following reasons. Firstly, this is a weak warming up of fluid in a top part of the region immediately behind the step for the range of  $x$  from 0.5 to 3. Although the presence of two eddies behind the step has to cause intensive heat exchange as it was in Figure 4. Secondly, this is a downstream fall of temperature in the right top part of the channel for  $10 < x < 20$ . For the given boundary conditions on the bottom B1 and top B3 walls this is possible only in the case of presence of the source term which plays the role of heat outflow in the energy equation. But such a term in equation (4) is absent. Hence, in these cases the boundary conditions (9)–(10) act as some kind of artificial heat outflow.

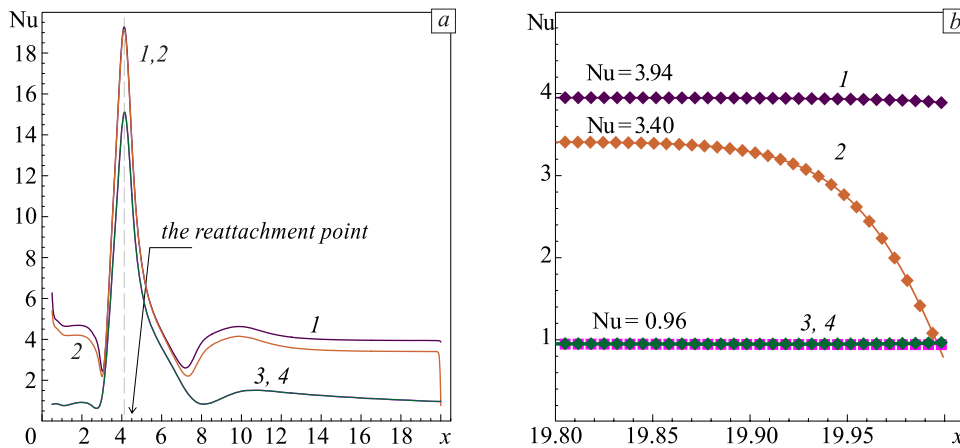
Due to the relative smallness of mass flow rate  $Q$  for  $h_c = 0.9$  (as was mentioned above,  $Q = 1 - h_c$  in dimensionless presentation) the mean velocity of stream for this case is less than the one for the previous case  $h_c = 0.5$ . Therefore, fluid downstream here has to warm up more quickly. These are the patterns of warming up of the liquid one can see in Figures 7, d–e. Whereas the fields of the heat transfer in Figures 7, b–c have structures similar to the patterns of isotherms in Figure 4, i.e., flow separation into the cold top part and the warm bottom one takes place. It turns out that despite of decrease in the flow velocity, heat continues to remain in a narrow lower region of



**Fig 7.** Fields of the fluid flow and heat transfer for  $ER = 10$ . (a) Pattern of the streamlines. Patterns of the isotherms for different boundary conditions at B6: (b) condition (9), (c) condition (10), (d) condition (11), (e) condition (12). Levels of  $\theta$  are: from 0.95 to 0.05 with decrement of 0.05, and 0.001

the channel. Such a picture of warming up of the fluid flow contradicts common sense. As a result, the solutions in Figures 7, b–c cannot be considered to be correct enough from the point of view of physical legitimacy of the process. Here it is necessary to pay attention to one very important feature of the consideration: the unsuccessful open boundary conditions (9)–(10) have distorted the solutions in the whole calculation domain, but not just in a small outlet boundary region.

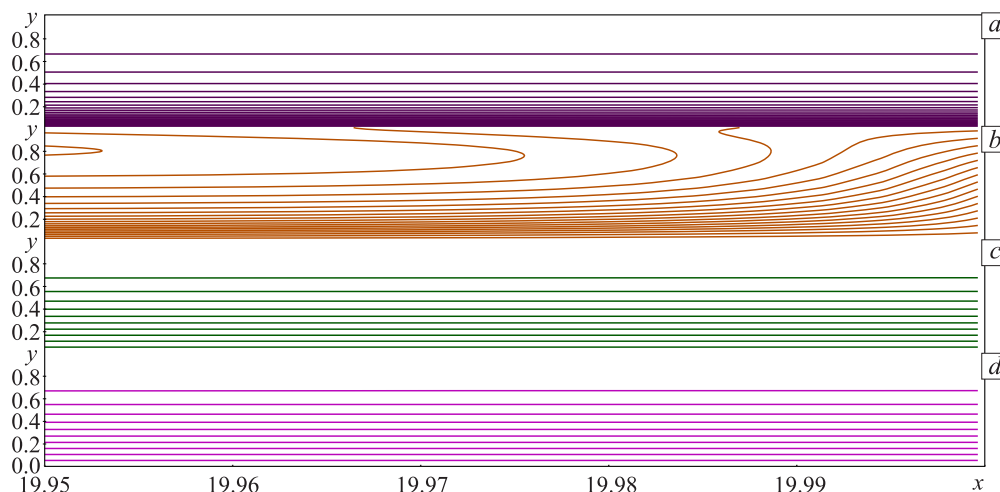
The profiles of Nusselt number at heated wall B1 for the whole length of the channel and for the very small part near the outlet channel border are observed in Figure 8.



**Fig 8.** Profiles of Nusselt number for  $ER = 10$  for different boundary conditions at B6: (a) the whole length of the domain downstream behind the step, (b) near outflow boundary of the domain. 1 – condition (9), 2 – condition (10), 3 – condition (11), 4 – condition (12)

It is obvious that for the reasons discussed above, Nu-profiles for conditions (9)–(10) (curves 1, 2) differ from such profiles for conditions (11)–(12) (curves 3, 4). It is not difficult to understand that due to a closer position of the cool layers of the liquid to the heated wall B1 curves of Nu 1 and 2 are situated higher than curves 3 and 4. The curve 2 (condition (10)) has a sharp downturn close to the outlet border due to the same reasons as for the case  $ER = 2$  because these reasons do not depend on the height of the backward facing step.

The detailed patterns of isotherms in the outlet part of the channel in Figure 9 explain the behaviors of Nu profiles in Figure 8, *b*. Really, the dense arrangement of the isotherms in Figure 9, *a* near the bottom wall is indicative of a thin warming up layer, therefore, the heat flux from the wall to the fluid flow has to be considerable (curve 1 in Figure 8, *b*). Very curved isotherms in the narrow region close to the outlet border (see Figure 9, *b*) suggest a sharp change of Nu value in this region (curve 2 in Figure 8, *b*). And finally, the rare plane-parallel isotherms in Figures 9, *c–d* correspond to a moderate constant Nusselt number (curves 3 and 4 in Figure 8, *b*).



**Fig 9.** Patterns of the isotherms near the outflow boundary of the domain for  $ER = 10$  for different boundary conditions at B6: (a) condition (9), (b) condition (10), (c) condition (11), (d) condition (12). Levels of  $\theta$  are: from 0.95 to 0.05 with decrement of 0.05

The analysis carried out above allows a conclusion that conditions (11)–(12) are more acceptable than conditions (9)–(10). But both  $ER$  cases have a common specific feature, namely, a plane-parallel (or almost plane-parallel) structure of the stream near the outlet border of the channel (see Figures 4, *a* and 7, *a*). The natural question arises: what will be the solution of the energy equation if the flow at the exit from the channel is not a plane-parallel one?

To obtain the answer on this matter, the following problem has been solved. From the whole domain of fluid flow data ( $0 \leq x \leq 20$ , see Figure 7, *a*) a subset of the flow data has been extracted ( $0 \leq x \leq 8$ , see Figure 10, *a*) for which the energy equation (4) was solved for two kinds of open boundary conditions (11) and (12) at the outlet border of the restricted domain. The results of these solutions are demonstrated in Figures 10, *c–d*. At the same time in Figure 10, *b* an input part of the isotherms field which is an extraction from the solution for the whole length of the channel (see Figure 7, *d*), is presented.

It is not difficult to see that the heat fields in Figures 10, *b–d* coincide well, whereas solutions in Figures 10, *b–c* differ essentially. Hence, of the two open boundary conditions (11) and (12), nonlinear condition (12) is a more preferable one. It should be noted that the deformations of the solution in Figure 10, *c* are very similar to the ones in Figures 7, *b–c*. This means that the reasons which have generated these deformations are the same.

Before making a final choice of the open boundary condition, it makes sense to explore the results of application to the heat problem of open conditions which belong to the class of so-called radiation methods. In particular, the well-known ‘Orlanski-type’ [2–4, 7, 40] open boundary conditions belong to this class. In the present investigation an implicit variant of two-layer Orlanski condition [4] and two-layer modification of simplified Orlanski method [3] are examined. These conditions in relation to the given study are written as follows:

– on the base Han et al. [4]

$$\theta_n^{k+1}(1 + c(\tau/\Delta x) - c(\tau/\Delta x)\theta_{n-1}^{k+1}) = \theta_n^k, \quad (13)$$

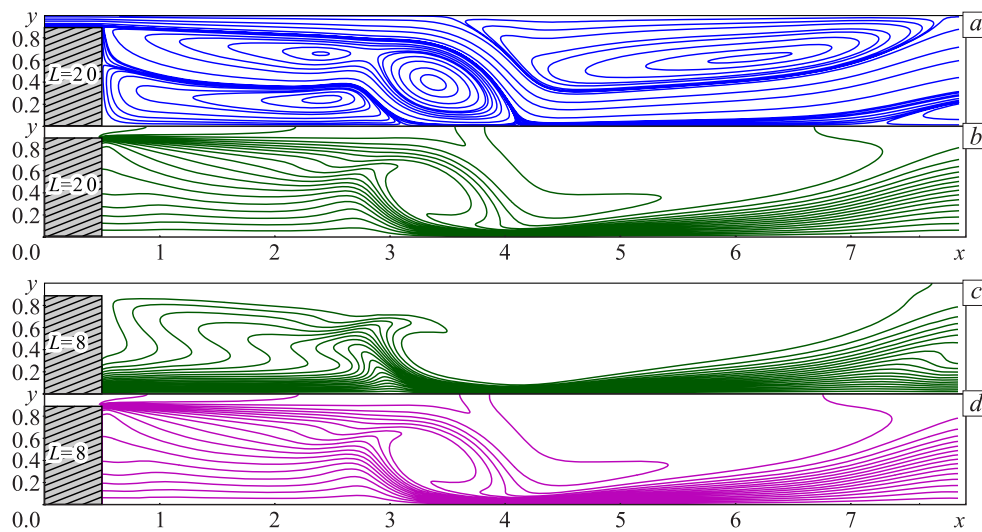
where

$$c = \begin{cases} \tilde{c}, & 0 \leq \tilde{c} \leq \Delta x/\tau, \\ \Delta x/\tau, & \tilde{c} > \Delta x/\tau, \\ 0, & \tilde{c} < 0; \end{cases}$$

– on the base Camerlengo et al. [3]

$$\theta_n^{k+1} = \begin{cases} \theta_{n-1}^k, & \tilde{c} \geq 0, \\ \theta_n^k, & \tilde{c} < 0. \end{cases} \quad (14)$$

Here  $\tilde{c} = -(\Delta x/\tau)(\theta_{n-1}^k - \theta_{n-1}^{k-1})/(\theta_{n-1}^k - \theta_{n-2}^k)$ ,  $n$  – the number of grid nodes along the  $x$ -coordinate,  $\Delta x$  – a grid step.



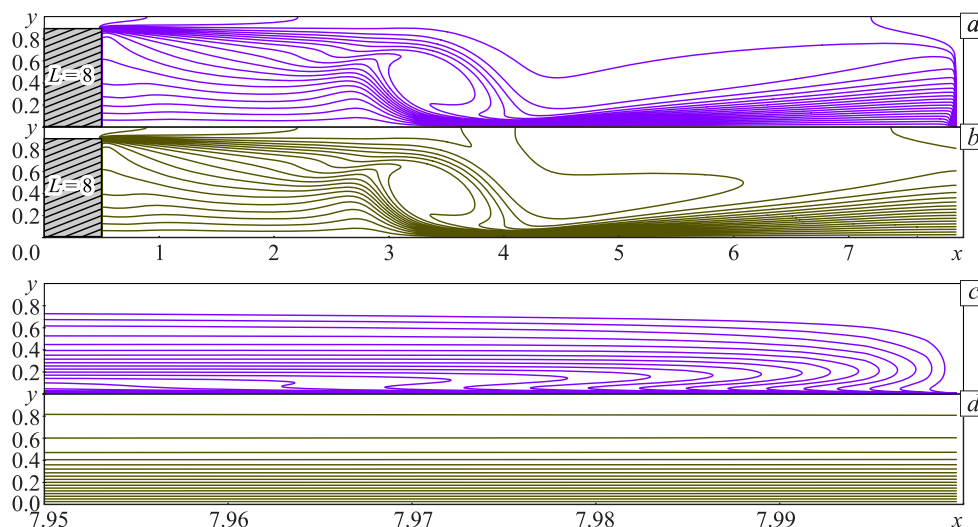
**Fig 10.** Fields of the fluid flow and heat transfer for  $ER = 10$ . (a) Pattern of the streamlines. Patterns of the isotherms for different boundary conditions at B6 and lengths of the domain: (b) condition (11),  $L = 20$ ; (c) condition (11),  $L = 8$ ; (d) condition (12),  $L = 8$ . Levels of  $\theta$  are: from 0.95 to 0.05 with decrement of 0.05

The patterns of isotherms for the problem just considered (see Figure 10) but for the outlet boundary conditions (13) and (14) both for the whole length of the calculation domain and for the narrow zone at the outlet border B6 are presented in Figure 11. First of all one can see that these patterns coincide with the heat field for condition (12) as a whole (compare Figure 10, *d* with Figures 11, *a–b*). Especially the pattern of isotherms for condition (14) almost completely repeats the one for condition (12), whereas the pattern for condition (13) at the outflow border has more significant differences (see the top right part of the pattern in Figure 11, *a* and *c*).

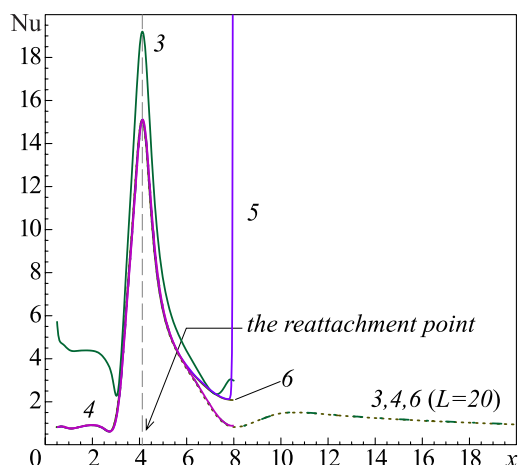
Nevertheless, there are differences between solutions for conditions (12) and (14) and it is well visible in Figure 12 where Nusselt number profiles for different kinds of open boundary conditions and lengths for the channel are demonstrated.

It is not difficult to see that all Nu profiles for the channel with length  $L = 20$  are the same and do not depend on the open boundary conditions (dashed curves). Hence, this dashed profile can be regarded as a true solution of the problem both for the whole length of the channel and for the region  $0 \leq x \leq 8$ . Distinctions between the Nu-profile for condition (12), on the one hand, and the curves for conditions (13) and (14), on the other hand, begin at a distance of 15–20 % from the output section of the ‘short’ channel with  $L = 8$  (curves 5 and 6). Moreover, profile 5 ‘shoots up’ at the exit from the channel because of the non-physical behavior of isotherms there (see Figure 11, *c*).

However, it should be noted that differences of the solutions for conditions (13) and (14) (curves 5 and 6) from the correct solution under condition (12) (curve 4) are much less than the differences between the solutions for conditions (11) and (12) (curves 3 and 4). Especially that distinctions between solutions for ‘radiation-type’ boundary conditions (13), (14) and nonlinear one (12) are



**Fig 11.** Patterns of the isotherms for different boundary conditions at B6. For the whole length of the domain: (a) condition (13), (b) condition (14); at the outflow boundary of the domain: (c) condition (13), (d) condition (14). Levels of  $\theta$  are: from 0.95 to 0.05 (a)–(b), and to 0.15 (c)–(d) with decrement of 0.05



**Fig 12.** Profiles of Nusselt number for  $ER = 10$  for different boundary conditions at B6 and the domain lengths: solid lines —  $L = 8$ , dashed lines —  $L = 20$ ; 3 — condition (11), 4 — condition (12), 5 — condition (13), 6 — condition (14)

localized in rather a small region at the exit from the channel. Hence, at a good choice of a channel length a solution in the similar region can just be neglected.

## § 5. Conclusions

The solutions of the 2D laminar steady incompressible flow and heat transfer over backward-facing step flow test problem for the short channel have been studied. The objective of this study was numerical investigation of the effect of the open boundary conditions on the solution of the heat transfer problem. The numerical technique of splitting of the physical factors and the authors' original method for solving the systems of linear algebraic equations with sparse matrix were used to obtain solutions of the problem. The test numerical results obtained within this research are in a very good agreement with those presented in the literature.

All calculations had been carried out for the following parameters of the problem statement:  $Re = 1000$ ,  $Pr = 0.71$ . During the investigation such problem parameters as the channel length, the backward facing step height, and the kind of open boundary condition at the outlet border of



the channel were varied: the channel length  $L = 8, 10,$  and  $20$ ; the expansion ratio  $ER = 1.5, 2,$  and  $10$ . On the whole, six kinds of open boundary conditions were examined. It is possible to draw the following conclusions on the basis of analysis of the obtained solutions in the considered range of change of the problem parameters and the kinds of open boundary condition:

1. For the expansion ratio  $ER = 10$  the solution of Navier–Stokes equations has a complex circulation structure. In particular, immediately behind the step two large eddies are situated one above the other. Moreover, one of them has two centers of rotation and a saddle point between them.
2. An unsuccessful open boundary condition can substantially distort the heat field in the whole calculation domain, and not just in a narrow zone near the output border.
3. The nonlinear condition (12) is the most preferable to stationary heat problems from the considered variants of open boundary conditions.

## REFERENCES

1. Roache P.J. *Computational fluid dynamics*, Albuquerque: Hermosa Publ., 1976, 446 p.
2. Orlanski I. A simple boundary condition for unbounded hyperbolic flows, *J. Comput. Phys.*, 1976, vol. 21, issue 3, pp. 251–269. DOI: [10.1016/0021-9991\(76\)90023-1](https://doi.org/10.1016/0021-9991(76)90023-1)
3. Camerlengo A.L., O'Brien J.J. Open boundary conditions in rotating fluids, *J. Comput. Phys.*, 1980, vol. 35, issue 1, pp. 12–35. DOI: [10.1016/0021-9991\(80\)90031-5](https://doi.org/10.1016/0021-9991(80)90031-5)
4. Han T.Y., Meng J.C.S., Innis G.E. An open boundary condition for incompressible stratified flows, *J. Comput. Phys.*, 1983, vol. 49, issue 2, pp. 276–297. DOI: [10.1016/0021-9991\(83\)90127-4](https://doi.org/10.1016/0021-9991(83)90127-4)
5. Sani R.L., Gresho P.M. Resume and remarks on the open boundary condition minisymposium, *Internat. J. Numer. Methods Fluids*, 1994, vol. 18, issue 10, pp. 983–1008. DOI: [10.1002/fld.1650181006](https://doi.org/10.1002/fld.1650181006)
6. Ol'shanskii M.A., Staroverov V.M. On simulation of outflow boundary conditions in finite difference calculations for incompressible fluid, *Internat. J. Numer. Methods Fluids*, 2000, vol. 33, issue 4, pp. 499–534. DOI: [10.1002/1097-0363\(20000630\)33:4<499::AID-FLD19>3.0.CO;2-7](https://doi.org/10.1002/1097-0363(20000630)33:4<499::AID-FLD19>3.0.CO;2-7)
7. Blayo E., Debreu L. Revisiting open boundary conditions from the point of view of characteristic variables, *Ocean Modelling*, 2005, vol. 9, issue 3, pp. 231–252. DOI: [10.1016/j.ocemod.2004.07.001](https://doi.org/10.1016/j.ocemod.2004.07.001)
8. Dong S., Karniadakis G.E., Chrysosostomidis C. A robust and accurate outflow boundary condition for incompressible flow simulations on severely-truncated unbounded domains, *J. Comput. Phys.*, 2014, vol. 261, pp. 83–105. DOI: [10.1016/j.jcp.2013.12.042](https://doi.org/10.1016/j.jcp.2013.12.042)
9. Kawamura T., Tanaka S., Mabuchi I., Kumada M. Temporal and spatial characteristics of heat transfer at the reattachment region of a backward-facing step, *Experimental Heat Transfer*, 1987, vol. 1, issue 4, pp. 299–313. DOI: [10.1080/08916158708946348](https://doi.org/10.1080/08916158708946348)
10. Sparrow E.M., Kang S.S., Chuck W. Relation between the points of flow reattachment and maximum heat transfer for regions of flow separation, *International Journal of Heat and Mass Transfer*, 1987, vol. 30, issue 7, pp. 1237–1246. DOI: [10.1016/0017-9310\(87\)90157-8](https://doi.org/10.1016/0017-9310(87)90157-8)
11. Nakamura H., Takaki S., Yamada S. Spatio-temporal characteristics of heat transfer in separated and reattaching flows, *Proc. ASME-JSME-KSME 2011 Joint Fluids Engineering Conference: Volume 1, Symposia – Parts A, B, C, and D*, 2011, ASME, pp. 3977–3988. DOI: [10.1115/AJK2011-25020](https://doi.org/10.1115/AJK2011-25020)
12. Nakamura H., Yamada S. Quantitative evaluation of spatio-temporal heat transfer to a turbulent air flow using a heated thin-foil, *International Journal of Heat and Mass Transfer*, 2013, vol. 64, pp. 892–902. DOI: [10.1016/j.ijheatmasstransfer.2013.05.006](https://doi.org/10.1016/j.ijheatmasstransfer.2013.05.006)
13. Aung W., Baron A., Tsou F.-K. Wall independency and effect of initial shear-layer thickness in separated flow and heat transfer, *International Journal of Heat and Mass Transfer*, 1985, vol. 28, issue 9, pp. 1757–1771. DOI: [10.1016/0017-9310\(85\)90149-8](https://doi.org/10.1016/0017-9310(85)90149-8)
14. Kondoh T., Nagano Y., Tsuji T. Computational study of laminar heat transfer downstream of a backward-facing step, *International Journal of Heat and Mass Transfer*, 1993, vol. 36, issue 3, pp. 577–591. DOI: [10.1016/0017-9310\(93\)80033-Q](https://doi.org/10.1016/0017-9310(93)80033-Q)
15. Valencia A., Hinojosa L. Numerical solutions of pulsating flow and heat transfer characteristics in a channel with a backward-facing step, *Heat Mass Transf.*, 1997, vol. 32, issue 3, pp. 143–148. DOI: [10.1007/s002310050104](https://doi.org/10.1007/s002310050104)
16. Kanna P.R., Das M.K. Conjugate heat transfer study of a two-dimensional laminar incompressible wall jet over a backward-facing step, *Journal of Heat Transfer*, 2006, vol. 129, issue 2, pp. 220–231. DOI: [10.1115/1.2424235](https://doi.org/10.1115/1.2424235)

17. Tihon J., Pěnkavová V., Havlica J., Šimčík M. The transitional backward-facing step flow in a water channel with variable expansion geometry, *Experimental Thermal and Fluid Science*, 2012, vol. 40, pp. 112–125. DOI: [10.1016/j.exptthermflusci.2012.02.006](https://doi.org/10.1016/j.exptthermflusci.2012.02.006)
18. Yang Y.T., Huang M.L. Numerical studies of heat transfer characteristics by using jet discharge at downstream of a backward-facing step, *Acta Mech.*, 1998, vol. 128, issue 1–2, pp. 29–37. DOI: [10.1007/BF01463157](https://doi.org/10.1007/BF01463157)
19. Tsay Y.-L., Chang T.S., Cheng J.C. Heat transfer enhancement of backward-facing step flow in a channel by using baffle installation on the channel wall, *Acta Mech.*, 2005, vol. 174, issue 1–2, pp. 63–76. DOI: [10.1007/s00707-004-0147-5](https://doi.org/10.1007/s00707-004-0147-5)
20. Hong B., Armaly B.F., Chen T.S. Laminar mixed convection in a duct with a backward-facing step: the effects of inclination angle and Prandtl number, *International Journal of Heat and Mass Transfer*, 1993, vol. 36, issue 12, pp. 3059–3067. DOI: [10.1016/0017-9310\(93\)90034-4](https://doi.org/10.1016/0017-9310(93)90034-4)
21. Abu-Hijleh B. Convection heat transfer from a laminar flow over a 2-D backward facing step with asymmetric and orthotropic porous floor segments, *Numerical Heat Transfer. Part A: Applications*, 1997, vol. 31, issue 3, pp. 325–335. DOI: [10.1080/10407789708914040](https://doi.org/10.1080/10407789708914040)
22. Batenko S.R., Terekhov V.I. Friction and heat transfer in a laminar separated flow behind a rectangular step with porous injection or suction, *J. Appl. Mech. Tech. Phys.*, 2006, vol. 47, issue 1, pp. 12–21. DOI: [10.1007/s10808-006-0002-7](https://doi.org/10.1007/s10808-006-0002-7)
23. Abu-Nada E., Al-Sarkhi A., Akash B., Al-Hinti I. Heat transfer and fluid flow characteristics of separated flows encountered in a backward-facing step under the effect of suction and blowing, *Journal of Heat Transfer*, 2007, vol. 129, issue 11, pp. 1517–1528. DOI: [10.1115/1.2759973](https://doi.org/10.1115/1.2759973)
24. Saha S., Mamun A.H., Hossain Z., Islam S. Mixed convection in an enclosure with different inlet and exit configurations, *Journal of Applied Fluid Mechanics*, 2008, vol. 1, no. 1, pp. 78–93.
25. Teruel F.E., Fogliatto E. Numerical simulations of flow, heat transfer and conjugate heat transfer in the backward-facing step geometry, *Mecánica Computacional*, 2013, vol. 32, no. 39, pp. 3265–3278.
26. Gada V.H., Datta D., Sharma A. Analytical and numerical study for two-phase stratified-flow in a plane channel subjected to different thermal boundary conditions, *International Journal of Thermal Sciences*, 2013, vol. 71, pp. 88–102. DOI: [10.1016/j.ijthermalsci.2013.03.022](https://doi.org/10.1016/j.ijthermalsci.2013.03.022)
27. Mitsoulis E., Malamataris N.A. Free (open) boundary condition: some experiences with viscous flow simulation, *Internat. J. Numer. Methods Fluids*, 2012, vol. 68, issue 10, pp. 1299–1323. DOI: [10.1002/fld.2608](https://doi.org/10.1002/fld.2608)
28. Papanastasiou T.C., Malamataris N., Ellwood K. A new outflow boundary condition, *Internat. J. Numer. Methods Fluids*, 1992, vol. 14, issue 5, pp. 587–608. DOI: [10.1002/fld.1650140506](https://doi.org/10.1002/fld.1650140506)
29. Dimakopoulos Y., Karapetsas G., Malamataris N.A., Mitsoulis E. The free (open) boundary condition at inflow boundaries, *J. Non-Newton. Fluid Mech.*, 2012, vol. 187–188, pp. 16–31. DOI: [10.1016/j.jnnfm.2012.09.001](https://doi.org/10.1016/j.jnnfm.2012.09.001)
30. Patankar S.V. *Numerical heat transfer and fluid flow*, New York: Hemisphere Publishing Corporation, 1980, 197 p.
31. Belotserkovskii O.M., Gushchin V.A., Shchennikov V.V. Use of the splitting method to solve problems of the dynamics of a viscous incompressible fluid, *USSR Computational Mathematics and Mathematical Physics*, 1975, vol. 15, issue 1, pp. 190–200. DOI: [10.1016/0041-5553\(75\)90146-9](https://doi.org/10.1016/0041-5553(75)90146-9)
32. Fomin A.A., Fomina L.N. Numerical simulation of viscous incompressible fluid in a short plane channel with backward-facing step, *Matem. Mod.*, 2016, vol. 28, no. 5, pp. 32–46 (in Russian). <http://mi.mathnet.ru/eng/mm3728>
33. Fomin A.A., Fomina L.N. Acceleration of the line-by-line recurrent method in Krylov subspaces, *Vestn. Tomsk. Gos. Univ. Mat. Mekh.*, 2011, no. 2 (14), pp. 45–54 (in Russian). <http://mi.mathnet.ru/eng/vtgu189>
34. Armaly B.F., Durst F., Pereira J.C.F., Schönung B. Experimental and theoretical investigation of backward-facing step flow, *J. Fluid Mech.*, 1983, vol. 127, pp. 473–496. DOI: [10.1017/S0022112083002839](https://doi.org/10.1017/S0022112083002839)
35. Chiang T.P., Sheu T.W.H., Fang C.C. Numerical investigation of vortical evolution in a backward-facing step expansion flow, *Appl. Math. Model.*, 1999, vol. 23, issue 12, pp. 915–932. DOI: [10.1016/S0307-904X\(99\)00019-0](https://doi.org/10.1016/S0307-904X(99)00019-0)
36. Erturk E. Numerical solutions of 2-D steady incompressible flow over a backward-facing step, Part I: High Reynolds number solutions, *Comput. & Fluids*, 2008, vol. 37, issue 6, pp. 633–655. DOI: [10.1016/j.compfluid.2007.09.003](https://doi.org/10.1016/j.compfluid.2007.09.003)
37. Rogers S.E., Kwak D. An upwind differencing scheme for the incompressible Navier–Stokes equations, *Appl. Numer. Math.*, 1991, vol. 8, issue 1, pp. 43–64. DOI: [10.1016/0168-9274\(91\)90097-J](https://doi.org/10.1016/0168-9274(91)90097-J)

38. Lewis R.W., Nithiarasu P., Seetharamu K.N. *Fundamentals of the finite element method for heat and fluid flow*, John Wiley & Sons, 2004, xiv + 341 p. DOI: [10.1002/0470014164](https://doi.org/10.1002/0470014164)
39. Cruchaga M.A. A study of the backward-facing step problem using a generalized streamline formulation, *Communications in Numerical Methods in Engineering*, 1998, vol. 14, issue 8, pp. 697–708. DOI: [10.1002/\(SICI\)1099-0887\(199808\)14:8<697::AID-CNMI55>3.0.CO;2-0](https://doi.org/10.1002/(SICI)1099-0887(199808)14:8<697::AID-CNMI55>3.0.CO;2-0)
40. Røed L.P., Smedstad O.M. Open boundary conditions for forced waves in a rotating fluid, *SIAM Journal on Scientific and Statistical Computing*, 1984, vol. 5, issue 2, pp. 414–426. DOI: [10.1137/0905031](https://doi.org/10.1137/0905031)

Received 01.07.2017

Fomin Aleksandr Arkad'evich, Candidate of Physics and Mathematics, Senior Researcher, Kuzbass State Technical University, ul. Vesennaya, 28, Kemerovo, 650000, Russia.

E-mail: [fomin\\_aa@mail.ru](mailto:fomin_aa@mail.ru)

Fomina Lubov' Nikolaevna, Candidate of Physics and Mathematics, Associate Professor, Kemerovo State University, ul. Krasnaya, 6, Kemerovo, 650000, Russia.

E-mail: [lubafomina@mail.ru](mailto:lubafomina@mail.ru)

**А. А. Фомин, Л. Н. Фомина**

**Численное решение задачи теплопереноса в коротком канале с обратным уступом**

**Цитата:** Вестник Удмуртского университета. Математика. Механика. Компьютерные науки. 2017. Т. 27. Вып. 3. С. 431–449.

**Ключевые слова:** несжимаемое течение жидкости, теплоперенос, выходное граничное условие.

УДК 532.516.5, 536.24

DOI: [10.20537/vm170311](https://doi.org/10.20537/vm170311)

В статье рассматривается модельная задача несжимаемого течения жидкости и переноса тепла в коротком плоском канале с обратным уступом. Цель работы состоит в исследовании влияния граничного условия для потока тепла (температуры) на выходе из канала на характеристики теплопереноса внутри канала. Система уравнений Навье–Стокса и баланса тепла решаются численно с использованием равномерной сетки разрешением  $6001 \times 301$  узлов. Для разностной аппроксимации пространственных производных используется метод контрольного объема второго порядка. Достоверность получаемых решений подтверждена для широкого диапазона числа Рейнольдса ( $100 \leq Re \leq 1000$ ) и числа Прандтля  $Pr = 0.71$  путем сравнения с экспериментальными и теоретическими результатами, найденными в литературе. Анализируются картины течения, поля изотерм перегрева потока и поведение локального числа Нуссельта вдоль нагретой нижней стенки канала в зависимости от выбора выходного граничного условия для потока тепла (температуры). Показано, что этот выбор может оказать существенное влияние на характер прогрева течения внутри всего канала. По результатам исследования выбор сделан в пользу нелинейного граничного условия.

Поступила в редакцию 01.07.2017

Фомин Александр Аркадьевич, к. ф.-м. н., старший научный сотрудник, Кузбасский государственный технический университет имени Т. Ф. Горбачёва, 650000, Россия, г. Кемерово, ул. Весенняя, 28.

E-mail: [fomin\\_aa@mail.ru](mailto:fomin_aa@mail.ru)

Фомина Любовь Николаевна, к. ф.-м. н., доцент, Кемеровский государственный университет, 650000, Россия, г. Кемерово, ул. Красная, 6.

E-mail: [lubafomina@mail.ru](mailto:lubafomina@mail.ru)