Disturbing effects of a cylinder-form macro-roughness on the turbulent free-convection boundary layer: Large Eddy Simulation

A M Levchenya, E V Kolesnik and E M Smirnov

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

E-mail: levchenya_am@spbstu.ru

Abstract. The paper presents results of eddy-resolving numerical simulation of non-isothermal turbulent flow in the vicinity of an adiabatic circular cylinder that disturbs the turbulent free-convection boundary layer developing along a vertical heated plate. The computational domain used includes (i) a relatively long section, where the upstream free-convection boundary layer goes over to its natural state from the model state prescribed at the inlet boundary with a synthetic turbulence generator, (ii) a mid-section covering the disturbing cylinder, mounted normally on the plate, (iii) an outlet section that is much longer than the cylinder diameter. The mathematical model is based on the Navier-Stokes equation written with the Boussinesq approximation. In the case simulated, the height of the cylinder (equal to its diameter) is about two times less than the approaching boundary layer half-width, if estimated from the velocity profile. Characteristic features of unsteady phenomena in the flow disturbed by the macro-roughness are analyzed with a focus on behavior of arising horseshoe-like vortex structures. Quantitative data on heat transfer augmentation near the cylinder is presented.

1. Introduction

Dynamic and heat exchange processes developing in free-convection boundary layers play a dominant role for performance of a number of devices. The base configuration for studying of these processes is a vertical heated plate. In some cases, the free-convection boundary layer forming near a vertical heated surface can be disturbed by a local obstacle. Examples are structural elements of an industrial plant or a residential building, containers for storing spent nuclear fuel, etc.

If a free-convection boundary layer interacts with a 3D obstacle, conditions favorable for formation of horseshoe-shaped vortices are created. Previously [1], we considered the problem of the interaction of a turbulent free-convection boundary layer with a semi-infinite cylinder that penetrates the layer completely. According to the steady-state solution obtained on the base of the Reynolds-averaged Navier-Stokes equations, RANS, the horseshoe-shaped vortex structures forming near the cylinder lead, in particular, to sharp peaks in the local heat transfer rate. It is known, however, that in the related case of turbulent forced-convection boundary layer interacting with a bluff body, large-scale eddies occurring just upstream of the body-plate junction are essentially non-stationary, and this circumstance affects the wall heat flux pattern significantly (see, for example, [2]).

The present paper covers results of Large Eddy Simulation, LES, of turbulent flow and heat transfer in the vicinity of an adiabatic finite-height circular cylinder that is inserted into the turbulent...
free-convection boundary layer developing along a vertical heated plate. Computations were carried out using ANSYS Fluent in version 16.2.

2. Problem formulation and computational setting

2.1. Description of the case
The flow configuration considered is illustrated in Figure 1a. It consists of a single adiabatic circular cylinder of diameter \( d \) and height \( h \) that disturbs the turbulent free convection boundary layer developing along a vertical heated plate. The latter is kept at constant temperature \( T_w \), the ambient temperature \( T_a \) is less than \( T_w \).

![Figure 1.](image)

In case of \( (T_w-T_a)/T_a << 1 \), the local state of the approaching turbulent boundary layer, assuming statistically two-dimensional, can be characterized by the local Grashof number, \( Gr_\delta = g \beta_T (T_w - T_a) \delta^3 / \nu^2 \). Here \( g \) is the gravity acceleration, \( \beta_T \) is the thermal expansion coefficient, \( \nu \) is the fluid kinematic viscosity. The used local integral thickness of the layer, \( \delta \), is defined as integral of \( u/\bar{u}_{max} \) across the boundary layer to \( \delta_T \)-position, where \( u \) and \( \bar{u}_{max} \) are time-averaged values of local and maximum streamwise velocity (x-velocity) in a considered section of the boundary layer, \( \delta_T \) is the thermal boundary layer thickness defined as a normal-to-plate coordinate where the fluid temperature differs from the ambient temperature by 1% of \( (T_w - T_a) \).

Let’s assume that the cylinder is inserted into the boundary layer at a position where \( \delta = \delta^* \) in case of no cylinder. As a result, two determining parameters of the considered problem can be introduced: the Grashof number \( Gr_\delta^* \) based on \( \delta^* \), and a parameter being the ratio of the boundary layer thickness and the cylinder diameter, \( \beta^* = \delta^*/d \). Extra two determining parameters are the Prandtl number, \( Pr \), and a relative height of the cylinder, \( h/d \). The present work covers the case of \( h/d = 1, Pr = 0.7, \beta^* \approx 2.1, \) and \( Gr_\delta^* \approx 3 \times 10^6 \).

In fact, calculations were carried out using dimensional flow parameters. Thermal boundary conditions were taken according to the experimental study of air convection reported by Tsuji and Nagano [3]: \( T_w = 60^\circ C, T_a = 16^\circ C \). As in [3], physical properties of air were taken at the mean temperature, equal to 38\(^\circ\)C, except that the thermal expansion coefficient was evaluated at \( T=T_a \).

The three-dimensional computational domain used (Figure 1a) has a form of a parallelepiped with the inserted cylinder of 40 mm diameter. The x-axis of a Cartesian system is directed vertically upward, and xz-planes are parallel to the plate. The computational domain size is 2.1x0.40x0.48 m in x-, y- and z-direction correspondingly. The cylinder axis is positioned in the domain middle (vertical) plane, at a distance of 1.4 m from the inlet section.

2.2. Mathematical model and numerics
Basically, the flow simulated is described by the Navier-Stokes equations added by the energy equation. Physical properties of the fluid are assumed constant, and the Boussinesq approach is used to
describe the buoyancy effect. Eddy-resolving LES technique is used in the present simulation. The Wall-Adapting Local Eddy-Viscosity (WALE) model [4] was chosen as the subgrid-scale model. The turbulent Prandtl number, Prₜ, was taken as 0.85.

The boundary conditions are defined as follows. The no-slip condition is prescribed at the plate (1 in Figure 1a) and at the cylinder surface (2). As noted above, the plate surface temperature is kept constant, and the cylinder is treated as adiabatic. Profiles of mean velocity components, temperature and two turbulence parameters corresponding to a model 2D turbulent free-convection boundary layer are prescribed at the inlet section (3). These profiles have been obtained from 2D RANS-computations of the free-convection boundary layer carried out with the k-ε RNG model. Inlet velocity fluctuations, needed for LES calculation, are generated applying the Vortex Method, available in the ANSYS Fluent code. No temperature fluctuations are introduced at the inlet section. For the outlet section (4), the complex inflow/outflow (“pressure-outlet”) condition is used. The “pressure-inlet” condition is applied at the external permeable boundary (5) parallel to the plate. Periodicity conditions are imposed at the boundaries confining the computational domain in the spanwise direction (6, 7).

Three-dimensional quasi-structured multiblock grid used for computations was generated by means of translating a 2D xz-plane grid along the y-direction. The source 2D grid consisted of H- (main domain) and O-structure (around the cylinder) blocks. The points of translation were clustered near the plate to provide a proper value of the average normalized distance from the centre of the first computational cell to the wall, y* is about 0.2. A special attention was paid to achieve a good grid resolution in the leading-edge region, as well as near the cylinder top (Figure 1b). Longitudinal and spanwise grid spacing is quite suitable for LES application: <|x'|>||z'||<10. Overall size of the computational grid is of about 16 million cells.

A second-order time-accurate backward scheme was applied for the implicit physical-time advancing. The NITA solver with the Fractional-Step option was used running the code. The second-order central scheme was applied for evaluation of convective fluxes, both for the momentum and energy equations. The time advancing was carried out with a step of 0.004 seconds that corresponds to 0.2 nondimensional units, t_scale=h_b/ut_b, where h_b=w/ut_b is a buoyancy length scale and ut_b=[gβ(T_w-T_a)v]^{1/3} is a buoyancy velocity. A sample of duration more than 6000t_scale has been calculated for getting flow statistics after a transient period.

3. Results and discussion

In order to define inlet boundary conditions, preliminary 2D computations of the free-convection boundary layer developing on a long vertical heated plate were carried out using the RANS k-ε RNG model. The streamwise distributions obtained with this model for the wall skin friction coefficient and the Nusselt number have shown a smooth transition from the dependence corresponding to the laminar boundary layer to that typical for turbulent flow regime. From the other side, experimental data [3] allows one to conclude that the region of more or less developed turbulence in the boundary layer lays at the Grashof numbers (based on the distance x from the plate lower edge) exceeding Gr_x≈10^{10}. Taking all this into account, it was decided to use profiles u(y), T(y), k(y) and ε(y) given by the 2D RANS solution at Gr_x≈10^{10} as the mean-flow inlet boundary conditions for the 3D case simulated. As mentioned above, the inlet turbulent content was produced by the Vortex Method synthetic turbulence generator.

First LES-based computations were performed for the case of no cylinder, in order to define a length of the region, where the turbulent free-convection boundary layer goes over to its natural state starting from the model state prescribed at the inlet boundary. Figure 2 illustrates process of development of vortex structures peculiar to the turbulent free-convection boundary layer (simulated also in our recent contribution [5]).

Figure 3 presents the computed distribution of the time- and span-averaged Nusselt number, Nu*=q_wh_b/λ(T_w-T_a), along the plate. Here q_w is the wall heat flux, λ is the thermal conductivity. As seen, strong influence of the artificial inlet boundary conditions is limited by a region of about 0.5 m length. One can approximately adopt that the boundary layer has gone over to its natural state at a length of about 1 m.
Figure 2. Vortex structure of the free-convection boundary layer computed: isosurfaces of the Q-criterion colored by streamwise velocity.

Figure 3. Streamwise variation of the time- and span-averaged Nusselt number along the plate in case of no disturbing cylinder.

Time- and span-averaged profiles of velocity and temperature extracted from the LES solution at a position of 1.2 m ($Gr_\theta=2.3\times10^6$) are compared in Figure 4 with the measurement data [3]. Here and hereinafter $\theta=(T-T_a)/(T_w-T_a)$, and $\zeta=-y(\partial\theta/\partial y)|_{y=0}$ is dimensionless normal coordinate. A good agreement between the predicted and the experimental profiles can be considered as an additional justification of the fact that the boundary layer achieves a natural state before coming to the cylinder positioned at a distance of 1.4 m from the computational domain inlet section.

Figure 4. Comparison of the computed profiles of the mean (a) velocity and (b) temperature with experimental data [3]: a section in the undisturbed region upstream of the cylinder.

Then, the LES-based computations were carried out for the configuration with the cylinder inserted into the boundary layer. Figure 5 illustrates instantaneous distributions of the $x$-velocity, normalized with $u_b$ and the reduced temperature over the mid-plane of the computational domain. As well, isolines of time-averaged values of these flow parameters are given in the plot.

Figure 5. Instantaneous distributions of the dimensionless (a) $x$-velocity and (b) temperature over the mid-plane; isolines of time-averaged values are plotted as well.

In Figure 5 one can clearly see a separation zone adjacent to the back side of the cylinder. However, the boundary layer separates also from the plate upstream of the cylinder, where a system of unsteady horseshoe-shaped vortex structures is formed. Due to conditions of substantially unsteady flow in the approaching boundary layer, these junction flow structures dramatically change their form...
in time. Figure 6 shows typical instantaneous flow patterns observed in the mid-plane just upstream of the plate-cylinder junction. “Footprints” of the horseshoe-shaped unsteady vortex structures on the plate are illustrated in Figures 7 in combination with instant maps of the wall heat flux $h$ normalized with the mean value, $h_0$, evaluated in the case of no cylinder.

At certain instances, the system of horseshoe-shaped vortex structures computed exhibits behavior similar to that observed in the experiments with the turbulent forced-convection boundary layer interacting with a bluff-body [2]. Let’s consider the instants illustrated in Figures 6 and 7: (a) the fluid inrushes to the plate due to intense action of the main horseshoe-shaped vortex; (b) there occurs a secondary horseshoe-shaped vortex that has a reverse direction of rotation. Both these cases are characterized by heat transfer intensification in a narrow zone near the leading edge of the cylinder. From time to time, the saddle point shifts noticeably from the mid-line (Figure 7c), and sometimes even the main horseshoe-shaped vortex can disappear, so that only a turn of the flow upstream of the obstacle is observed (Figure 6d). However, in any case, the intensified motion of the fluid inside the zone occupied by the horseshoe-shaped vortex structures reduces the thermal boundary layer thickness that results in a significant heat transfer augmentation in this area.

![Figure 6](image1.png)

**Figure 6.** Typical unsteady flow patterns just upstream of the plate-cylinder junction: mid-plane streamline patterns are over-imposed onto corresponding temperature contours.

![Figure 7](image2.png)

**Figure 7.** Typical instant distributions of the plate heat flux in the vicinity of the cylinder; surface streamline patterns are plotted as well.

Time variations of the wall heat flux at two points positioned on the mid-line upstream of the cylinder is shown in Figure 8. Positions of the points are defined by the normalized distance $x' = (x_{LE} - x)/d$ from the cylinder leading edge. One can see that close to the obstacle, $x' = 0.05$, the heat flux fluctuations are extremely high. Figure 9 presents mid-line variations of the wall-shear stress and the heat flux upstream of the cylinder leading edge, data for both the time-averaged values and the standard deviations are given. Here, the wall-shear stress, $\tau_w$, is also referred to the mean value, $\tau_{w,0}$ predicted in case of no cylinder. Remarkably that the region of considerable deviations of wall friction from the undisturbed-layer value is several times wider as compared to the region of wall heat flux variations. This fact has been previously revealed in the RANS-based simulation [1]. Generally, the grade of increasing the wall shear stress and the heat transfer predicted with LES for the region affected by the horseshoe-shaped vortices is of the same order as that estimated with the RANS model.
Figure 8. Time variations of the wall heat flux at two monitoring points positioned on the mid-line upstream of the cylinder: (1) $x' = (x_{LE} - x)/d = 0.05$ and (2) $x' = 0.825$.

Figure 9. Mid-line variations of (a) wall-shear stress and (b) heat flux upstream of the cylinder: (lines) time-averaged values and (vertical bars) standard deviations.

4. Conclusions
Large eddy simulation of non-isothermal air turbulent flow and heat transfer near a 3D obstacle disturbing the turbulent free-convection boundary has been carried out. Model boundary conditions imposed at the computational domain inlet section are constructed on the base of a 2D RANS solution added by a synthetic turbulent content for velocity fluctuations. Temperature fluctuations at the inlet can be set zero since they arise and grow quickly downstream. A length of the section needed the free-convection boundary layer to go over to its natural turbulent state has been evaluated.

Because of interaction of the turbulent free-convection boundary layer with a finite-length cylinder mounted on a heated vertical plate, a complicated 3D flow field develops upstream of the cylinder-plate junction, where a system of horseshoe-shaped vortex structures, dramatically changing in time, is formed. Highly intensified motion of the fluid inside the zone occupied by the horseshoe-shaped vortex structures reduces the thermal boundary layer thickness that leads to a very significant heat transfer augmentation and to growth of wall shear stress in this area. Peak values are several times higher, if compared to the undisturbed boundary layer case.

Acknowledgments
The study is supported by the Russian Scientific Foundation under grants no. 18-19-00082. The results of the work were obtained using computational resources of Peter the Great Saint-Petersburg Polytechnic University Supercomputing Center (www.spbstu.ru).

References
[1] Levchenya A M, Smirnov E M, Zhukovskaya V D and Ivanov N G 2018 Int. Heat Transf. Conf.-16 (Beijing) vol 14 (Begell House) pp 5493–500
[2] Praisner T J and Smith C R 2006 J. Turbomach. 128 pp 747–54
[3] Tsuji T and Nagano Y 1988 Int. J. Heat Mass Transfer 31 pp 1723-34
[4] Nicoud F and Ducros F 1999 Flow, Turbulence and Combustion 62 (3) pp 183–200
[5] Smirnov E M, Abramov A G, Smirnovsky A A, and Smirnov P E 2018 J.Phys.: Conf.Ser. 1128 012090