Compressibility effect on heat transfer intensified by horseshoe vortex structures in turbulent flow past a blunt-body and plate junction

E V Kolesnik, A A Smirnovsky and E M Smirnov
Department of Fluid Dynamics, Combustion and Heat Transfer, Peter the Great St.Petersburg Polytechnic University, 29 Polytechnicheskaya Street, 195251, St.Petersburg, Russia
E-mail: kolesnik_ev@mail.ru

Abstract. Results of numerical simulation of turbulent heat and mass transfer in the 3D problem of flow past a blunt-body and plate junction are presented. Comparison with the known experimental data is given; solution grid sensitivity is discussed. Numerical solutions obtained for incompressible fluid flows with different Prandtl numbers, and for the compressible gas flow with the Mach number ranging from 0.01 to 0.5 are analyzed. The effect of compressibility on heat transfer intensification in the region occupied by horseshoe-shaped vortex structures is investigated.

1. Introduction
The leading-edge region of turbulent juncture flows is of particular interest in turbomachinery applications since the presence of horseshoe vortices has been found to significantly increase local endwall heat transfer rates [1]. For high-pressure turbines, these effects can cause thermal-mechanical fatigue, spalling of thermal barrier coatings, and airfoil endwall/platform burning. Consequently, experimental and numerical investigation of such phenomena is of great practical interest. Many studies, both experimental and numerical, have been devoted to the structure of horseshoe-shaped vortices that arise in such configurations and their effect on the heat transfer intensification [2-6]. Due to the complexity of conducting measurements in real gas turbines, model configurations are usually used. Extensive experimental work was conducted by Praisner and Smith [5], who studied turbulent flow past cylinder with a 5:1 trailing edge fairing mounted on a flat plate, with localized heating applied in the endwall region (figure 1), and obtained detailed heat transfer data.

Figure 1. Schematic view of the configuration considered.
The experiments were carried out in an open water channel. The Reynolds number of inflow, Re, based on the cylinder diameter $D = 0.15$ m, was equal to 24400. The thickness of the undisturbed turbulent boundary layer, $\delta$, which would have been reached to the place of the cylinder installation in case of its absence, was of $0.35D$. The part of the plate near the cylinder was heated by a constant heat flux, $q$. For these conditions, a detailed numerical simulation based on the RANS approach was carried out in [6]; the calculation results agree well with experimental data. It should be emphasized that the experiment [5] was performed for water, whereas for gas-turbine applications, the compressible gas flow is of primary interest. The present paper is aimed at the evaluation of different factors affecting the compressible gas flow in the given configuration.

For the incompressible case, the flow structure and heat transfer are defined by two dimensionless parameters: the Reynolds number and the Prandtl number, Pr. For the compressible flow case, the Mach number, M, should be specified as well. The major aim of the present numerical study is to reveal the influence of compressibility on the heat transfer in the given configuration. For this purpose, we firstly consider the solution obtained when setting all the parameters as in [5] (to compare CFD results with experimental data). Secondly, we evaluate the effect of the Prandtl number changing from 0.7 to 10, and then we consider solutions for the compressible gas flow with varying the Mach number from 0.01 to 0.5. For all calculations, $Re = 24400$, and the ratio of the incoming boundary-layer thickness and the blunt fin diameter $\delta/D = 0.35$. Calculations for compressible flow were performed for air (ratio of specific heats, $\gamma$, is equal to 1.4).

The computational domain and its geometric parameters/sizes are shown in figure 2. The flow is assumed to be symmetric, and therefore only half of the configuration was considered. The no-slip condition was imposed on the solid surfaces of the body-plate juncture. All surfaces were assumed adiabatic, except the part of the plate near the leading edge. There two types of boundary conditions – constant heat flux or isothermal wall condition – were considered. To define boundary conditions at the inlet section of the 3D computational domain, 2D turbulent flow developing on a flat plate was computed beforehand using the Menter SST model [9]. The inlet free stream turbulence intensity of 0.5% was specified, and the viscosity ratio was taken as 6.7. The constant value of static pressure was specified at the outlet.

![Figure 2](image-url)  
**Figure 2.** Computational domain: blunt-body (1), plate (2) with heated part (3), symmetry planes (4), inlet boundary (5), outlet boundary (6).

2. **Computational method and numerical aspects**

Unstructured finite-volume in-house code SINF/Flag-S, which is under development at the Department of Fluid Dynamics, Combustion and Heat Transfer of SPbPU, was used to perform the calculations. In the incompressible fluid case, the SIMPLE-like algorithm was applied for solving the Reynolds-averaged incompressible-fluid Navier-Stokes equations and the energy equation assuming constant physical properties of the fluid.

For compressible flow simulation, the Reynolds-averaged compressible-gas Navier-Stokes equations with the perfect gas equation of state were integrated using an implicit stepping method. The gas viscosity was assumed to be dependent on temperature according to the Sutherland’s law, taken
for air, and the heat conductivity was related to the viscosity through a constant Prandtl number. Convective fluxes were evaluated by the Roe scheme [7], an extension of which to the second-order was achieved via the MUSCL slope-limiting approach [8].

For turbulence modeling, the Menter SST model [9] was used, and in all the computations the turbulent Prandtl number was taken as 0.9.

It is well known that in case of low Mach number flows, the numerical methods developed originally for compressible flows become inefficient: system stiffness resulting from disparate acoustic and convective velocities (the matrix condition number becomes large) causes convergence rates to deteriorate [10]. To overcome this difficulty, the method of Weiss and Smith [11] has been implemented into SINF/Flag-S. In the case of very low Mach numbers the implemented numerical scheme is found to provide a possibility of getting converged solutions during the computer time, which is comparable or even lower than a run with the SIMPLE-like algorithm.

In order to examine the quality of calculations, several hexahedral meshes were used. For the case considered, the main characteristic of the mesh used is the ratio $D/(\Delta x)^*$, where $(\Delta x)^*$ is the average cell size in X-direction in the region, where the main horseshoe vortex axis intersects the X-axis. Three meshes were considered: Mesh 1 with $D/(\Delta x)^* = 100$ (0.7 mln cells), Mesh 2 with $D/(\Delta x)^* = 200$ (1.7 mln cells), and Mesh 3 with $D/(\Delta x)^* = 400$ (6.1 mln cells).

The results of the present work were obtained using computational resources of Peter the Great Saint-Petersburg Polytechnic University Supercomputing Center (scc.spbstu.ru).

3. Results and discussion

3.1. Grid sensitivity and comparison with experiment (incompressible fluid case)

Figure 3a gives a general view of the incompressible flow calculated with the most refined mesh (of ~ 6.1 mln cells) at $Pr = 6.3$. As it is clearly visible, the major peculiarities of the velocity field are attributed to the leading edge endwall region. When a turbulent boundary layer approaches a blunt-body obstruction, the adverse pressure gradient creates a three-dimensional separation, which reorganizes the vorticity accumulated in the boundary-layer into a leading-edge horseshoe vortex. The separation point locates upstream of the leading edge at a distance exceeding the edge radius.

Figure 3b shows the symmetry-plane endwall Stanton number distributions obtained using different meshes in comparison with experimental data [5] and numerical data [6]. Stanton number is calculated as follows:

$$St = q_w / (\rho V_C_p \Delta T).$$

Here, $\Delta T$ is evaluated using inlet and wall temperatures: $\Delta T = T_w - T_{in}$.

Figure 3. (a) Flow structure: velocity map, surface streamlines and Stanton number map; (b) Stanton number distribution along the symmetry line on the plate.
In accordance with the experiments, the computations predict two peaks in the Stanton number distribution. Far away from the body, the computed Stanton number values fit experimental data for all meshes. The Stanton number in the region adjacent to the leading edge is underestimated. The global St peak computed is located closer to the leading edge, as compared with the experiments, and it increases with better grid resolution. In general, the results of the calculations are in good agreement with the measurement data. It can be noted also that the results are in a very good agreement with the numerical data reported in [6]. Distinctions between the results obtained with Mesh 2 and Mesh 3 are relatively small. All the results presented below are obtained using Mesh 3.

3.2. The Prandtl number effect
The Prandtl number influence on the heat transfer rate was studied for the incompressible flow case. For \( Pr \) ranging from 0.7 to 10, figure 4 shows the computed Stanton number distributions along the symmetry line, as well as the symmetry-plane \( X \)-velocity field with an imposed streamline pattern. In the picture given, one can see the main horseshoe-shaped vortex, the secondary, tertiary vortex preceding it, and another relatively weak vortex upstream. A corner vortex is formed near the intersection of the plate and the leading edge of the body. Note, that each of the calculated St-distributions contains several local maxima that correspond to stagnation points between vortices.

It is known that the Stanton number evaluated for a turbulent boundary layer developing along a flat plate is proportional to \( Pr^{-2/3} \). To reveal the Prandtl number influence on the heat transfer peculiarities in the region occupied by the horseshoe-shaped vortices, figure 4 shows the values of the Stanton number divided by \( Pr^{-2/3} \). In general, while maintaining a similarity of the local heat transfer patterns for different Prandtl numbers, an increase in \( Pr \) leads to the growth of the peak values of \( St/Pr^{-2/3} \). Remarkably, an additional local maximum is distinguished (at \( X/D = -0.22 \)) in the case of \( Pr \geq 3 \), which is not predicted at smaller Prandtl numbers.

![Figure 4](image)

**Figure 4.** Symmetry-line Stanton number distributions calculated with different Prandtl number values.
3.3. Compressibility effects

According to the experiment [5], the results presented above were obtained by imposing a constant heat flux on a part of the plate. For a compressibility effect study related to gas turbine applications, it is more reasonable to impose the constant temperature boundary condition, with a wall temperature, $T_w$, lower than that for the inlet gas, $T_{in}$. Figure 5 presents a comparison of results obtained with two types of boundary conditions, BC, for the incompressible flow and for the compressible gas case with $M = 0.01$. In the latter case, the wall heat flux $q = 10.5 \text{ W/m}^2$ specified for the first type of boundary condition and the difference between the inlet and wall temperatures used for the second BC variant, $T_w/T_{in} = 0.983$, were chosen to be relatively small in order to prevent large variations of gas properties due to temperature change.

In the case of essentially compressible gas flow, the effect of viscous heating cannot be neglected, contrary to what is usually assumed for incompressible flow calculations. Since this effect leads to gas heating within the boundary layer the Stanton number is usually evaluated using the adiabatic wall temperature, which is calculated by the boundary layer law as follows:

$$T_{aw} = T_w \left(1 + \left(\frac{\gamma - 1}{2} r M^2\right)\right), \quad r = Pr^{1/3} \quad \text{recovery factor.} \quad (2)$$

Accordingly, in the Stanton number definition (1), the temperature difference is evaluated as $\Delta T = T_w - T_{aw}$.

As illustrated in figure 4a, changes in the wall boundary conditions lead to relatively small distinctions in Stanton number distributions. For the constant wall temperature, the peak values of St are slightly larger. Note that the compressible flow with $M = 0.01$ can be effectively treated as incompressible fluid motion, and, as expected, the results obtained with two solvers are practically identical. For further calculations with the compressible gas formulation the constant wall temperature was used for the part of the plate marked by 3 in figure 2.

Figure 5b illustrates the effect of an increase in Mach number on the Stanton number distribution. One can see that the heat transfer pattern remains similar to the incompressible flow case up to $M = 0.3$, with a small growth of the separation region length. For $M = 0.5$, some changes in the flow pattern are observed. In particular, one additional horseshoe vortex appears upstream, as shown in figure 6, where a Mach number map and a streamlines pattern for the symmetry plane are presented. This leads to a notable Stanton number change in the region $-0.6 < X/D < -0.4$. Besides, all the vortices become larger and shift away from the body, and the separation region length increases.

Figure 5. Stanton number distributions: (a) effect of thermal boundary condition on the plate and (b) Mach number effect.
4. Conclusions

Numerical study of the 3D flow field and end-wall heat transfer in the juncture of a blunt-body and a flat plate has been performed both for incompressible fluid and compressible gas (air). The mesh-independent computational results obtained for the incompressible case have been compared with the experiment of [5]. The results of the present calculations are in good agreement with the experimental data.

In all the cases considered, the interaction of the incoming turbulent boundary layer with the symmetrical blunt-body mounted on the plate leads to the forming of a three-dimensional separation region with a system of multiple horseshoe vortices. Vortex structure of the flow in the separation zone defines remarkable peculiarities of heat transfer on the plate, in particular, the Stanton number has two pronounced local peaks.

The Prandtl number effect on the plate heat transfer has been evaluated. It has been revealed that an increase in the Prandtl number leads to a growth of peaks in the local Stanton number distribution.

To investigate the influence of gas compressibility on the heat transfer, solutions for different Mach numbers, in the range of 0.01 to 0.5, have been obtained. Compressibility is shown to noticeably affect the heat transfer pattern, especially at rather high Mach numbers (M > 0.3): horseshoe vortices become larger and shift away from the body, leading to the growth of peak Stanton numbers; the separation region length increases as well.

References

[1] Kang M, Kohli A and Thole K A 1999 J. Turbomach. 121(3) 558-68
[2] Devenport W J and Simpson R L 1990 J. Fluid Mech. 210 23-55
[3] Fleming J L, Simpson R L et al. 1993 Experiments in Fluids 14(5) 366-78
[4] Ballio F and Franzetti S 2000 Proceedings of Advances in Fluid Mechanics 255-64
[5] Praisner T J and Smith C R 2006 ASME J. Turbomach. 128(4) 755-762
[6] Levchenya A M, Smirnov E M and Goryachev V D 2010 International Journal of Heat and Fluid Flow 31(6) 1107-12
[7] Roe P L 1981 J. Comput. Phys. 43(2) 357-72
[8] Van Leer B. 1979 J. Comput. Phys. 32(1) 101-36
[9] Menter F R, Langtry R and Kuntz M 2003 Turbulence, Heat and Mass Transfer 4 625-32
[10] Turkel E 1987 J. Comput. Phys. 72(2) 277-98
[11] Weiss J M and Smith W A 1995 AIAA Journal 33(11) 2050-57