NUMERICAL SIMULATION OF LOCAL HEAT TRANSFER IN ROTATING TWO-PASS SQUARE CHANNELS

Alexander I. Kirillov*, Vladimir V. Ris*, Evgueni M. Smirnov** and Dmitry K. Zaitsev**
*Department of Thermoengineering
St.-Petersburg State Technical University, St.-Petersburg, 195251, Russia
**Department of Aerodynamics
St.-Petersburg State Technical University, St.-Petersburg, 195251, Russia

A two-pass rotating channel is a representative model for simulations of the conditions typical for internal serpentine cooling passages in gas turbine rotor blades. Local heat transfer in two-pass channels has been studied experimentally by several research groups for the last decade. Less knowledge is available for features of turbulent flow developing in the channel. Real flow and heat transfer phenomena are made very complicated by the presence of a sharp U-bend, and also by Coriolis and rotational buoyancy forces. The Coriolis-influenced flow field depends on the channel orientation with respect to the rotation axis and, consequently, the channel orientation significantly affects the heat transfer from the individual surfaces. Three-dimensional numerical simulation is expected to be an effective tool for prediction of the flow and local heat transfer distributions at various operating conditions. However, additional efforts are needed to assess whether engineering turbulence models are able to predict phenomena of such complexity.

The present contribution is devoted to numerical simulation of turbulent air flow and heat transfer developing in a two-pass square channel rotating in the orthogonal mode. Basically, conditions adopted at the experiments of Dutta & Han[1] are considered. The only difference consists in use of constant wall temperature conditions over the heated region instead of constant wall heat flux arranged in the measurements. As an example, Figure 1 shows the flow geometry for case when the turn direction is aligned with the rotation axis. For given orientation, the flow is determined by the Reynolds number, $\mathrm{Re}=W_mD/\nu$, the rotation number, $\mathrm{K}=\omega D/W_m$, the normalized mean radius, $R_m/D=(R_0 + L/2)/D$, and the temperature factor, $\varepsilon_T=(T_{wall} - T_{in})/T_{mean}$. Among the measurement data provided by Dutta & Han, runs with the Reynolds number of 5000 are of the major interest for the present study since the Coriolis and the buoyancy effects are the most pronounced ($\mathrm{K}=0.15$, $\varepsilon_T=0.13$, $R_m/D=55$). The heated area is $26D$ long.

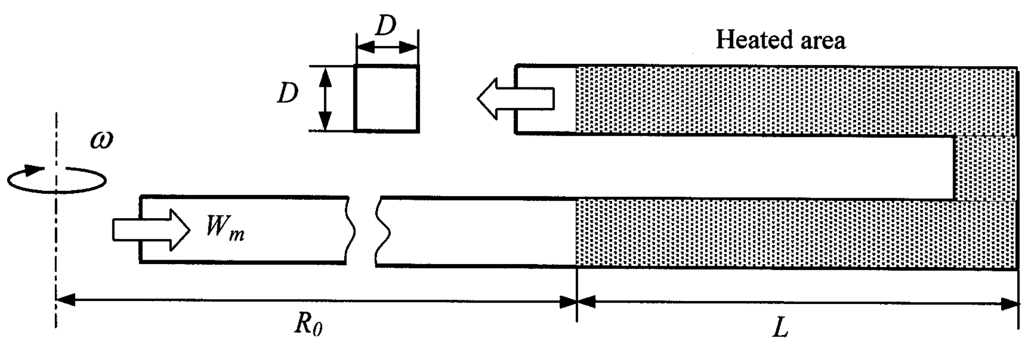

Figure 1. Flow geometry for case of parallel orientation of the turn with respect to the rotation axis

Governing equations are the full Reynolds-averaged Navies-Stokes equations written with the Boussinesq's approximation for incorporation of the centrifugal buoyancy. The energy equation is written in the form of static enthalpy balance. The governing equations of the standard high-Reynolds number k-ε model in combination with the wall function technique are used for turbulence modeling. It has been established that for Reynolds numbers of 5000 and two-three times higher the standard formulation of wall boundary conditions for the high-Re k-ε model results in a considerable over-prediction of heat transfer rate on any computational meshes sufficient for secondary flow resolution. Application of low-Re formulations is extremely computer-time consuming. To get more accurate results and less grid-sensitivity, modified wall functions suggested recently by one of the authors (E. Smirnov) are used for definition of turbulent parameters at the first computational point away of the wall.

The computational domain includes the heated region, an upstream section one hydraulic diameter long, and a downstream section of the same length. Adiabatic wall boundary conditions are imposed for both the upstream and downstream sections. The computational domain is covered by a 113x21x21 nonuniform mesh in case of the standard wall functions and by a 113x33x33 mesh in case of the modified wall functions that allow a closer distance of the first computational point to the wall.

The computations have been performed with a well-validated academic code (named SINF). This advanced 3D Navier-Stokes solver is based on the second-order finite-volume spatial discretization using the cell-centered variable arrangement and body-fitted block-structured grids. The artificial-compressibility method is used to link the velocity and pressure fields through the continuity equation. In case of an unsteady problem, the artificial-compressibility technique is applied at each physical time step. The pseudo-time stepping is performed with an effective implicit method. A QUICK-type upwind scheme is employed to compute convective fluxes on the stage of residual computation. The numerical dissipation introduced in the stabilizing operator of the left-hand side of the linearized governing equations is proportional to the spectral radius of the Jacobian matrices of the convective flux vectors.

A special section of the present contribution is devoted to the problem of inlet boundary condition definition. In the experiments of Dutta & Han[1] an upstream non-heated channel of about 40*D* long was assembled to provide the conditions of fully-developed isothermal flow at the inlet section of the heated region. For numerical simulation purposes an obvious and attractive way for inlet boundary condition definition is to compute beforehand characteristics of the fully-developed flow on the base of the formulation neglecting streamwise flow variations. It is known that such a simplified problem admits a mirror-symmetrical solution, however it can be unstable with respect to non-symmetrical perturbations. The problem formulated without assuming the mirror symmetry may have no steady state solutions. It just occurs for the chosen set of the Reynolds and the rotation numbers when the trailing surface is parallel to the rotation axis. In the present work the fully-developed isothermal flow has been computed on the base of the unsteady formulation used previously for laminar flow case[2]. The velocity and turbulent parameter fields averaged over the period of developing self-oscillations were used then as inlet boundary conditions for the two-pass channel non-isothermal flow problem.

As a representative example, Figure 2 compares some results for heat transfer obtained in the present work with data of Dutta & Han[1]. The new wall functions suggested lead to a much better agreement with the measurement results. Figure 3 shows complicated secondary-flow patterns and normalized temperature distributions. The effect of the sharp 180 deg turn is well pronounced.

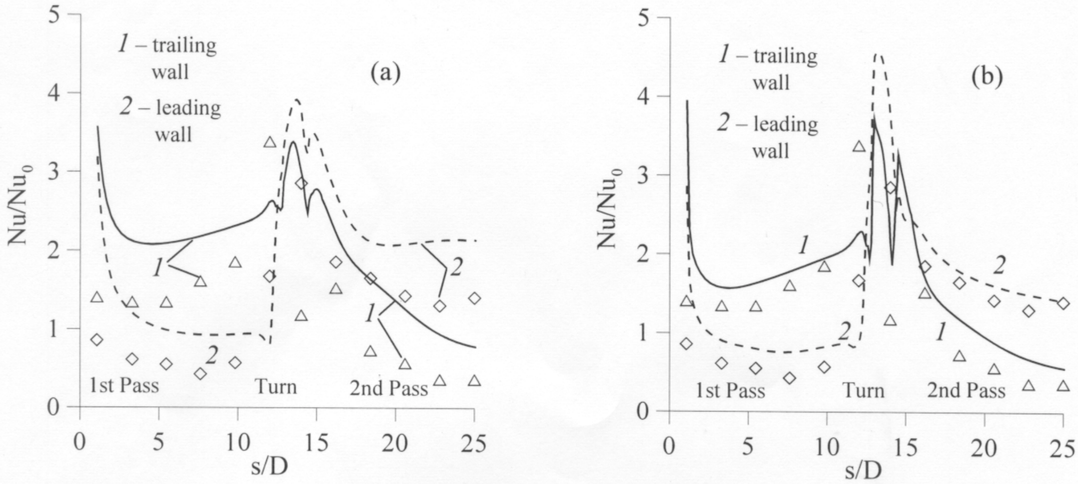

Figure 2. Variation of heat transfer rate with the distance along the channel centerline. Curves show results of present computations using (a) standard and (b) modified wall functions for k-ε turbulence model in comparison with (symbols) experimental data given by Dutta & Han[1] for Re=5000, K=0.15.

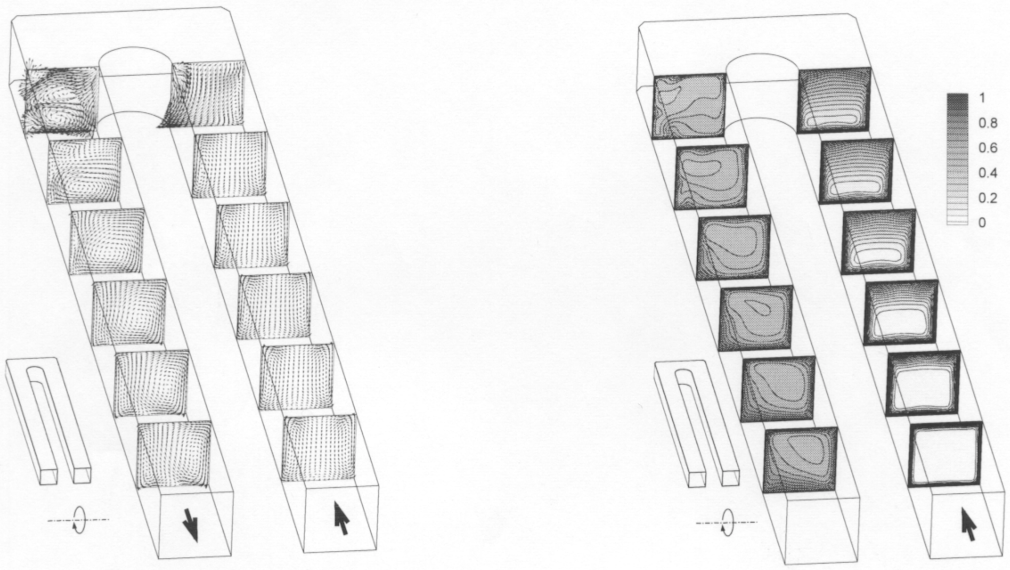

Figure 3. Secondary flow pattern and normalized temperature field.

To extract the centrifugal buoyancy effects, computations have been also performed with neglecting the buoyancy force. Further computations have shown that a change in the channel orientation causes a change in the secondary flow structure and in the distributions of turbulence parameters. Local heat transfer for the individual surfaces of the channel changes as well.

REFERENCES

1. Dutta, S. and Han, J.-C., Local Heat Transfer in Rotating Smooth and Ribbed Two-Pass Square Channels With Three Channels Orientations, *Trans. ASME, J. Heat Transfer*, Vol. 118, pp. 578-584, 1996.
2. Kirillov, A.I., Ris, V.V. and Smirnov E.M., Flow Bifurcation Aspects and the Prandtl Number Effect on Laminar Heat Transfer in a Long Duct Rotating in the Orthogonal Mode, *Proc.11th Int. Heat Transfer Conf.*, Kyongju, Korea, 1998, Vol. 3, pp. 45-50.