The turbulent flow structure in a wall jet blown through cylindrical holes into a transverse trench. Experiment and numerical simulation

I A Chokhar, A I Ochered’ko, M A Pakhomov, M V Philippov and V V Terekhov
Kutateladze Institute of Thermophysics SB RAS, Novosibirsk

E-mail: pakhomov@ngs.ru

Abstract. The results of experimental and numerical studies of the turbulent flow structure of a wall gas jet injected through round holes into a transverse trench are presented. It is shown that the main advantage of this method of coolant supply in comparison with the classical blowing through cylindrical holes in the protected surface is achieved at large blowing ratios. At that, the formation of vortex structures inside the trench ensures more uniform output of the secondary flow in the transverse direction. Comparative analysis with the data of our measurements indicates a quantitative correspondence between the results of the experiments and simulations.

1. Introduction
The continuous increase in the power of engines, various power equipment and units leads to an increase in the gas temperature at the turbine inlet [1,2]. Since the melting point of the materials used in the manufacture of turbine blades is significantly lower than the gas temperature, cooling of turbine blades is an urgent and important task. The hydrodynamic protection of the wall is the widespread cooling method. The coolant is blown into the boundary layer through cylindrical holes in the protected surface. The greatest positive effect is achieved by the use of shaped holes, special profiling of edges and other methods of modifying the channels or ducts geometry. One of such solution is the coolant supply through round inclined holes located in the trench [2]. The main advantage of this method is due to the fact that mixing of secondary coolant flow with the main flow of hot gas occurs in the trench. This leads to breaking the jets in the trench by large-scale vortex structures; so, it is not isolated jets but the secondary coolant flow in the form of a wall film that flows out of the trench. Therefore, the surface is protected not only directly behind the holes, but also in the transverse direction [2]. It is shown that the main parameters affecting the cooling efficiency are determined by the blowing ratio, the angle of inclination of the blown secondary flow, the shape of the edge of the outlet and the turbulence intensity and temperature of the main and secondary flows.

There are a number of studies of gas wall jets blown through round inclined holes, located in the transverse trench [2-4]. The work [2] reviews research on the subject and considers physical mechanisms influencing thermal efficiency of wall jets.

An experimental study of thermal efficiency at injection through inclined cylindrical holes in the transverse trench was carried out in [3]. The cooling efficiency significantly increased in comparison with well-known blowing through holes in the protected surface. Authors of [4] experimentally and
numerically investigated thermal efficiency of the gas wall jet, injected through inclined holes into the transverse trench. It was shown to be much higher than for classical injection, and an additional advantage of this method of coolant supply was achieved with large parameters of injection.

Almost all the experimental and numerical works on this subject found in the literature study only the thermal efficiency of wall jets. The analysis has not revealed any experimental studies of local mean and turbulent flow structures with coolant supply to the trench. It is quite obvious that it is the process of mixing of the main and secondary flows that determines a significant increase in thermal efficiency when blowing the wall jet into the trench. The aim of this work is to carry out experimental and numerical studies of the mixing process when a wall gas jet is blown through cylindrical inclined holes into a transverse trench.

2. Experimental setup
The measuring set-up (Fig. 1 and 2) consisted of a two-component LDA with adaptive time selection of orthogonal channels and a 3-component coordinate-positioning device. The secondary gas flow was fed through three holes with a diameter \( b = 3.2 \text{ mm} \), located in the trench. The distance between the axes of the holes was \( z = 10 \text{ mm} \). The length of the working test section was \( L = 950 \text{ mm} \), its width was \( 150 \text{ mm} \) and its height was \( H = 20 \text{ mm} \). Studies were performed for a single-row system of three holes arranged at an angle \( \alpha = 30^\circ \) (Fig. 3a), the depth of the trench was \( h = 2 \text{ mm} \) and its width was \( w = 6.4 \text{ mm} \) (Fig. 3b) \( (h/w = 0.375) \). The velocity of the main air flow was \( U_1 = 10 \text{ m/s} \), and velocity of the secondary flow at the output of the round holes was \( U_2 = 5 \text{ and } 10 \text{ m/s} \). In the first stage, the flow was studied for the cases of well-known classical flow injection through round holes without trenches and for the case with a trench.

![Figure 1](image1.png)

**Figure 1.** Experimental unit with a working area.

![Figure 2](image2.png)

**Figure 2.** Scheme of experimental unit: 1 – compressor, 2 – aerosol generator, 3 – fan, 4 – receiver-heater, 5 – supply tubes, 6 – LDA, 7 – working channel.

![Figure 3](image3.png)

**Figure 3.** Scheme of the flow without a trench (a) and with a trench \( h = 2 \text{ mm} \) (b).
3. Numerical model
For numerical study the authors used the LES and 3D RANS approaches. The LES method applied the WALE subgrid-scale model [4]. Simulations were performed on open CFD package OpenFOAM. RANS calculations were performed using a 3D mathematical model described in detail in [5]. The home code was used for our RANS simulations. Gas turbulence was modeled using a model of Reynolds stress component transport [6]. The elliptic blending Reynolds-stress model [7] was optimal to describe the complicated flow in the presence of separation regions and to correctly describe the mixing process in the trench. The numerical solution was obtained using the method of control volumes on staggered grids. The QUICK algorithm was used for convective terms of differential equations. Central differences were used for diffusion terms. Pressure field correction was realized with the finite-volume consistent SIMPLEC algorithm. RANS simulations were performed on a grid of $256 \times 100 \times 64$ (primary computation domain), $32 \times 10 \times 64$ (trench) and $32 \times 100 \times 64$ (the area above the trench). In total, the grid contained 1.86 million control volumes.

4. Results
The results are presented for the injection parameter $m = U_2/U_1 = 1$, $T_1 = T_2 = 293$ K and Reynolds number $Re = U_2h/\nu = 2000$. The profiles of mean longitudinal velocity are shown in Fig. 4 for two cases of wall jet injection without the trench (a) and with the trench with a depth $h = 2$ mm ($\delta h = 0.63$) (b). The symbols are the experimental results and solid and dashed lines show the results of numerical predictions with RANS and LES, respectively. Blowing the secondary flow into the main flow leads to a significant deformation of the gas velocity for both cases studied. For injection without a trench, numerical predictions show the presence of a pronounced wall jet located at some distance from the lower wall of the duct. The position of the maximum velocity of the secondary flow shifts from the duct wall as it moves downstream. The action of a wall jet manifests itself up to a distance $x/b = 6.3$ (see Fig. 4a). Then the wall jet mixes with the main flow and the velocity profile is leveled and begins to perform the laws of the developed flow in flat ducts [8]. For injection through holes in a trench, the gas velocity profile has a smoother shape and, on average, the gas velocities in the near-wall region of the channel are smaller in comparison with injection through inclined holes (see Fig. 4b). This can be explained by the fact that the main and secondary flows partially mix directly in the trench. The secondary flow does not leave the trench in the form of a discrete wall jet, but in the form of a gas “film”, which leads to a noticeable improvement in the protective properties of the film cooling in the transverse direction [2,3]. The qualitative agreement in the near-wall region of the results of numerical calculations for both approaches with the measurement data is observed. In the flow core region, good quantitative agreement is reached.

The comparison of the intensity fluctuations of the longitudinal velocity component for two cases of injection flows allows us to conclude that, in the near-wall region, behind the position of injection of the secondary flow, there is a zone with an increased level of turbulence in the mixing layer at $x/b = 0$ (see Figs. 5a and 5b). The intensity of pulsations significantly exceeds the corresponding value for the core of the flow. In the flow core, the intensity of turbulent velocity pulsations has almost the same value and good agreement is observed between the measured and numerical data.
Figure 4. Profiles of mean longitudinal velocity component along the channel length. Blowing through cylindrical holes without trench (a) and with the transverse trench of depth $h = 2$ mm (b). 1 – experiment, 2 – LES, 3 – RANS.
Conclusion
The paper presents data of numerical (LES, RANS) and experimental (LDA) study of flow structure of the wall gas jet, blowing through inclined round holes for the cases without trench and for the wall jet injection through circular holes in the transverse trench.

The experiment has shown that counter-rotating vortex pairs for trenchless blowing mix the flow in the channel with the jet flow much closer to the surface, which worsens thermal and aerodynamic efficiency, especially at long distances. The longitudinal velocity deformation for trench injection is much stronger. This suggests that the wall gas jet for injection into the trench is maintained at long distances. In general, there is a qualitative agreement between our experiment and simulations.
Acknowledgements
The study was financially supported by the Russian Foundation for Basic Research (grant no. 18-08-00386) (measurements and RANS results) and Ministry of Science and Higher Education (Project No. AAAA-A17-117030310010-9) (LES results).

References
[1] Volchkov E P 1983 Film Cooling (Novosibirsk: Nauka) [in Russian]
[2] Bunker R S 2005 ASME J. Heat Transfer 127 441
[3] Lu Y, Dhungel A, Ekkad S V and Bunker R S 2009 ASME J. Turbomach 131 011003 13
[4] Nicoud F and Ducros F 1999 Flow, Turbulence and Combust 62 183
[5] www.openfoam.com
[6] Pakhomov M A, Terekhov V I, Khalatov A A and Borisov I I 2015 Thermophys. Aeromech. 22 329
[7] Fadai-Ghotbi A, Manceau R and Boree J 2008 Flow, Turbulence and Combust 81 395
[8] Kutateladze S S, Leont'ev A I 1990 Heat Transfer, Mass Transfer and Friction in Turbulent Boundary Layers (Washington: Taylor&Francis Inc.)