Investigation of aerodynamics and heat transfer of the modular jet recuperator

P D Alekseev and Yu L Leukhin
Northern (Arctic) Federal University named after M.V. Lomonosov
Russia, 163002, Arkhangelsk, Severnaya Dvina Emb. 17
p.alekseev@narfu.ru

Abstract. A study of the aerodynamics and heat transfer of a jet modular recuperator with a change in its geometric characteristics has been carried out. The influence of the in-line and staggered arrangement of the blowing holes, as well as the diameter of the perforated pipe is considered. In all considered variants, the number of holes, their diameter and gas flow rate through the recuperator remained unchanged. Numerical modeling of the problem was carried out in a three-dimensional setting using the ANSYS Fluent 15.0 software package. It was found that with the in-line arrangement of the blowing holes, secondary flows are formed between their longitudinal rows in the form of swirling jets of opposite rotation directed towards the outlet section of the recuperative device, through which the main part of the heated air flows out. With the staggered arrangement of the blowing holes, the formation of spiral vortices is disturbed, the air flow is carried out along the entire cross section of the annular channel, increasing the drift effect of the flow on the impact jets, which leads to a decrease in the intensity of heat transfer and its uniformity along the length of the working surface. An increase in the diameter of the inner perforated pipe leads to a decrease in the drift effect of the cocurrent flow on the jets, an increase in the distribution uniformity of the heat flux along the length of the heat transfer surface, and an increase in the heat transfer coefficient.

1. Introduction
Modern trends in recuperator engineering, taking into account the most efficient use of production areas, require the creation of compact recuperators with high thermal efficiency and operational reliability. To a large extent, these requirements are met by modular recuperators, which are easy to operate, can be easily built into vertical and horizontal gas ducts, and have the ability to get any required heat transfer surface by selecting the required number of modules [1]. One of the ways to intensify convective heat transfer in recuperative devices is the jet leakage of the heat transfer gas onto the surface. A sharp intensification of heat transfer is achieved by a short length of the jet path due to significant turbulence of the near-wall gas layer at the blown surface [2]. The jet modular recuperator shown in figure 1 works as follows: cold air is fed into the inner tube 1, perforated with blowing holes, from which, in the form of an impact jets system, it is supplied to the inner surface of the heat transfer tube 2 and heats up. The module itself is installed in the horizontal gas duct 3 and is washed from the outside by the flue gases escaping from the power engineering installation.

The purpose of this work is to study the aerodynamics and heat transfer of a modular jet recuperator in order to develop a design with a higher thermal efficiency.
2. Research method
Numerical modeling of aerodynamics and heat transfer in a jet modular recuperator was carried out with a model, the geometrical dimensions of which corresponded to a real recuperator. The working length of the outer (heat transfer) pipe was 1389 mm, and its inner diameter $d_2$ was 100.5 mm. The outer diameter of the perforated insert was $d_1 = 60.5$ mm, its wall thickness was 5 mm, and the diameter of the blow holes was $d_0 = 3.5$ mm. The number of holes along the perimeter of the cross section was 5, and the number of rows was 33. The holes along the length had in-line arrangement. The temperatures of the air supplied to the device and of the outer pipe surface were 19.8°C and 100°C respectively, the air flow rate was 0.0055 m$^3$s$^{-1}$. The air flow rate and temperatures were chosen to verify the calculation results according to the experimental data obtained on the experimental model of the jet modular recuperator with similar geometric dimensions and boundary conditions [1]. In this work, we additionally investigated the options for the staggered arrangement of the blowing holes, as well as with an increased outer diameter of the perforated pipe up to 75.5 mm. In all considered variants, the number of holes and gas flow rate (hence, the velocity of the jet outflow) remained unchanged.

Mesh models were built in the ANSYS ICEM CFD software module, implemented by the boundary correction method (Octree) and are unstructured tetrahedral meshes with a maximum dimension of 8 million cells (a fragment of the grid for one of the jets is shown in figure 2). Calculations performed on grids of higher dimension did not lead to a noticeable change in the calculation results.

The study of the problem was carried out in a three-dimensional setting using the ANSYS Fluent 15.0 software package. The flow was described through Navier-Stokes equations of continuity and energy that are Reynolds-averaged. Reynolds equations closure is performed using both: a two-parameter turbulence model SST (Shear Stress Transport) $k$-$\omega$ streamlines curvature correction. To solve the problem, the following boundary conditions were used: 19.8 °C temperature and 0.0055 m$^3$s$^{-1}$ flow rate of air supplied.
to the inner perforated pipe of the device; 100 °C outer pipe surface temperature; the air pressure at the outlet from the device was equal to atmospheric. The outflow velocity of the jets and the Reynolds number, determined from this velocity and the diameter of the blowing holes, were equal to 34.6 m s⁻¹ and 8.06 ⋅ 10³, respectively. The air flow rate and temperature were chosen to verify the calculation results according to the experimental data obtained on the experimental model of the jet modular recuperator with similar geometric dimensions and boundary conditions [1].

3. Research findings

The modular recuperator aerodynamic features of the type under consideration are determined by the conditions of impact jets interaction, flowing from the inner perforated pipe with a concave cylindrical surface while one-sided flow diversion from an annular channel. Due to the high resistance of the blowing holes, the gas flow rates through them differ by a maximum of 7%, and a higher flow rate is observed through the holes located at the blind end. The jets coming out of this holes, hit the surface of the outer tube forming a set of unsteady vortex structures. The combination of these structures leads to the formation of secondary flows in the form of spiral vortices rotating in opposite directions. Figure 3 shows the streamlines projections onto the channel cross-section, and figure 4 shows the distribution of secondary vortex flows along the length of the annular channel along one row of holes. With the in-line arrangement of the blowing holes, two swirling jets of opposite rotation are formed between their longitudinal rows, directed towards the outlet section of the recuperative device. As the vortices move, their intensity increases due to the receipt of additional energy and mass from the jets located closer to the outlet. Through the formed secondary vortices, the main part of the heated air flows out of the module.

With an increase in the air flow rate to the outlet section of the annular channel and the intensity of the secondary vortex motion, the drift effect of the cocurrent flow on the impact jets increases. Figure 5 shows the change in temperature along the annular channel length (the flow out of the channel occurs on the left). Along the lower part of the cross-section, which coincides with the blowing holes axes, it can be seen that as the outlet is approached, the drift effect of the cocurrent flow gradually increases, and in the upper part, located between the rows of jets, the heated air transfer by secondary vortices to the surface of the inner pipe is observed. Due to the air flow rate increase and the constrained jets arrangement, not only the increase in the speed of the rotational movement occurs, but also their spinning.

![Streamline projections on the cross-sections of the annular channel.](image_url)
The nature of the distribution of the heat flux density represented in figure 6 shows that the most intense local heat transfer on the surface of the outer (heat-transferring) pipe is observed on its half, located closer to the blind end. As the flow moves towards the outlet, the jet action influence to the surface weakens. The maximum values of the heat transfer coefficient along the generatrix of the outer tube, located opposite the blowing holes, decrease towards the outlet section by about 40% (figure 7 line 1), and its minimum values on the same generatrix (between the jet axes), on the contrary, to the exit increase by about 30%. It should be noted that there is a significant non-uniformity of heat transfer along this typical line. On average, the maximum values exceed the minimum ones by 3.6 times.
Figure 7. Distribution of heat transfer coefficients along the length of the outer pipe: line 1 and line 2 - maximum and minimum values along the generatrix located opposite the holes; line 3 - on the generatrix located between the rows of holes; line 4 - surface averages. Points - experimental data.

Line 3 shows the change along the length of the heat flux density along the generatrix of the outer surface located between the rows of blow holes. Heat transfer in this area is the lowest, despite the fact that it slightly increases towards the exit under the influence of secondary vortices. Moreover, it is 5.7 times less than the average maximum value (line 1).

The change along the length of the surface-averaged heat transfer coefficient (line 4) and its comparison with the experimental data show satisfactory agreement.

With the staggered arrangement of the same number of blowing holes, the conditions for the interaction of the jets with the surface of the outer tube and the formation of secondary vortices change. The absence of "corridors" does not allow the formation of spiral vortices directed from the blind end to the exit. Air flow is carried out over the entire cross-section of the annular channel, increasing the drift effect on the jets. The distribution of the heat flux density over the surface of the outer pipe with a staggered arrangement of the blowing holes is even more uneven in length (figure 8) than with a in-line one. In this case, the heat transfer coefficient averaged over the surface is 9.7% less.

Figure 8. Distribution of the heat flux density over the surface of the outer tube with a staggered arrangement of the blowing holes.

With an increase in the diameter of the inner perforated pipe from 60.5 mm to 75.5 mm, the relative distance to the heat transfer surface \( \frac{d_h}{d_0} \) decreases by 37.5% \( (d_h = d_2 - d_1 \) is the hydraulic diameter of the annular channel). As a result, the drift effect of the cocurrent flow on the jets is reduced, and the uniformity of the heat flow distribution along the length of the heat transfer surface is increased (figure 9). The heat transfer coefficient averaged over the surface increases by 5%, while the resistance of the module increases by 10%.

Figure 9. Distribution of the heat flux density over the heat transfer surface with a perforated pipe diameter \( d_1 = 75.5 \) mm.

It should be noted that in all the variants considered in the work, there is a significant non-uniformity of heat transfer at the working surface, due to the local impact of jets on it.
4. Conclusions
With the in-line arrangement of the blowing holes, secondary flows are formed between their longitudinal rows in the form of swirling jets of opposite rotation, directed towards the outlet section of the recuperative device, through which the main part of the heated air flows out.

The staggered arrangement of the holes disrupts the formation of spiral vortices. Therefore, the airflow is carried out along the entire cross section of the annular channel, increasing the drift effect of the flow on the impact jets, which leads to a decrease in the intensity of heat transfer and its uniformity along the working surface length.

An increase in the diameter of the inner perforated pipe leads to a decrease in the drift effect of the cocurrent flow on the jets, an increase in the distribution uniformity of the heat flux along the length of the heat transfer surface, and an increase in the heat transfer coefficient.

References
[1] Saburov E, Ostashev S, Orekhov A, Leukhin Y, Nikiforov V, Kreinin E and Shaginyan I 1988 J. Industrial energy. 6 33
[2] Dyban E and Mazur A 1982 Convective heat transfer with jet cooling of bodies. (Kiev: NAUKOVA DUMKA)
[3] ANSYS 15 - User’s guide