Intensification of heat transfer in the inlet of the convective stage of the recuperative-burner unit

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

Abstract. The study of aerodynamics and heat transfer in the recuperator convective stage of the recuperative-burner unit with the jet leakage of the flue gas flow onto the outer cylindrical surface is carried out. Numerical modeling of the problem was carried out in a three-dimensional formulation using the ANSYS Fluent software package. It was found that in the original design of the recuperative-burner unit, there is a significant unevenness of heat transfer along the length and perimeter of the working surface of the convective stage. In the initial section of the annular gap, a stagnant zone with the lowest heat transfer rate is observed. To eliminate the stagnant zone and to intensify heat transfer on the surface in this area, it is proposed to make the entrance to the perforated pipe in the form of an inner quarter of a torus; to install smooth protrusions on its surface; to locate an annular flow divider on the inner surface of the heat transfer wall, which separates the front part of the annular channel with formation of a set of vortex chambers. The research results are presented.

1. Introduction

Power engineering installations are large consumers of high-calorie, expensive liquid or gaseous fuel, therefore, the task of increasing their efficiency is very important. Due to the high potential of the heat in the flue gas, the role of heat secondary use – recuperation – significantly increases. Utilization of flue gas heat from industrial furnaces of machine-building enterprises by means of modern highly efficient and compact design recuperative devices allow to save fuel energy resources by 21 - 34%, reduce air pollution and thereby contribute to solve the problems of environment protection [1].

Among the various types of recuperators, one of the most efficient, simple in design and relatively cheap devices are recuperative-burner units (RBU). They present the combination of typical burners with recuperators located in the burner settings close to each other [2, 3]. Figure 1 shows the cross-section of a recuperative burner unit operating as follows. The air supplied to the recuperator 2 through the nozzle 4 tangentially to the inner surface of the swirl generator 3 is spun, passes the straight air annular channel 5, and heats up from its inner surface - the heat transfer wall 11 of the flue gas channel.

Then the air is turned by 180° and through the return annular channel 6 and the collector 7 is directed to the burner 1. Through the inlet 12, high-temperature waste products of combustion enter first the radiation stage 8 of the flue gas channel, and then with a lower temperature go into the convective stage 9. In the radiation stage, the flue gas transfers heat to the air through the wall of the flue channel 11, mainly due to radiation, and in the convective stage - by convection due to the jet
leakage of flue gases from the perforated pipe 10 onto the wall 11. As a result of the jet leakage, the process of heat transfer to the air moving along the annular channel is intensified. This design can provide heating of the air used for fuel combustion up to 350 ... 400 °C at 750 ... 1200 °C flue gas temperature [4].

Figure 1. Cross-section of the recuperative burner unit

Heat transfer in the RBU annular channel between the swirling air flow and its surfaces has been studied in detail in [3 - 5]. It was found that the highest heat transfer rate on the inner surface of the annular air channel 5 is observed near the swirling flow generator 3. From the side of the flue gas, this part of the heat transfer wall is located in the convective part of the flue gas channel 9, which is blown by jets flowing from the holes of the perforated pipe 10.

An effective way to enhance heat transfer in heat exchange devices is the use of impact jets, that is, the creation of a system of heat-medium jets flowing onto the heat exchange surface. Heat transfer increases due to significant turbulence of the near-wall boundary layer of gas on the blown surface. The mechanism of heat transfer intensification is due to the loss of flow stability and additional generation of vortex structures [6-8]. Simple in design, jet cooling systems are widely used in various technology fields: aviation, space, energy, light and chemical industries, etc.

The problem considered in this work is complicated by the fact that the impact jets outflow occurs on the concave heat transfer surface. In this case, an asymmetric, one-sided removal of the heat-medium from the channel leads to a significant effect of the carry-over flow, as well as to the curvature of streamlines with the formation of complex vortex structures.

2. Research method
Numerical modeling of aerodynamics and heat transfer was carried out on a model of a recuperative burner block with 140 kW nominal thermal power. The flue gas duct (11 in figure 1) is a cylindrical pipe with 140 mm inner diameter and 930 mm working length, the radiation part of the duct being 480 mm, and the convective part of 450 mm. The perforated cylinder is made of a pipe with 104 mm inner diameter and 114 mm outer diameter. The cylinder surface has 204 holes of 8 mm diameter: 12 holes in each row along the perimeter and 17 rows lines with 30 mm pitch along the length of the cylinder.

The grid model was built in the ANSYS ICEM CFD software module, implemented by the boundary correction method (Octree). Since the problem under study can be considered as axisymmetric, for the
rational use of computational resources, the solution was carried out in one-quarter of the model (Figure 2). As a result, the grid model is an unstructured tetrahedral grid with a dimension of 16 million cells. For the near-wall boundary layer, the grid resolution $y^+ \approx 1$ is provided.

The research was carried out in a three-dimensional setting using the ANSYS Fluent 15.0 software package. The flow was described by the Navier-Stokes of continuity and energy equations that are Reynolds-averaged. The closure of the Reynolds equations was performed using the SST (Shear Stress Transport) $k$-$\omega$ two-parameter turbulence model with curve correction.

![Grid model of the studied part](image)

**Figure 2.** Grid model of the studied part

To solve the problem, the following boundary conditions were used: the temperature and speed of the air supplied to the radiation stage of the device equaled to $20 \, ^{\circ}C$ and $11.3 \, m/s$, respectively; outer pipe surface temperature $100 \, ^{\circ}C$; the air pressure at the outlet of the device equaled to atmospheric. The outflow velocity of the jets and the Reynolds number, determined from this velocity and from the diameter of the blowing holes, were $16 \, m/s$ and $8.5 \cdot 10^3$, respectively.

### 3. Research findings

It has been experimentally established that the pressure distributions along the length both inside the perforated cylinder 9 (Figure 1) and in the annular gap (channel) between it and the heat transfer wall 11 have significant unevenness. So in a perforated pipe, as the flow moves to its blind end, the pressure increases by about 15%, and in the outer annular gap, on the contrary, the pressure drops by about 7 times. This leads to the fact that the pressure drop between the internal volume of the perforated pipe and the annular gap in the holes located near the blind end is much greater than at the beginning of the pipe.

Figure 3 shows the distribution of the total velocity in the longitudinal cross-section of the convective stage of the recuperator. It should be noted that the outflow velocity of jets from the last rows of holes is about 4 times higher than from the first ones. Accordingly, the gas flow rate through the holes increases significantly in the direction of the blind end. Gas entering the annular channel from the front rows of holes forms a cocurrent flow, which has a blowing effect on the jets blown out near the blind end of the perforated pipe. Obviously, in order to increase the uniformity of the flow distribution over all holes, it is necessary to ensure the same pressure drop in them along the length of the perforated pipe.

As the flow enters the perforated pipe, a long region of separated flow is formed from its inlet, sharp edge. Low pressure in this area leads to the formation of a return flow from the annular channel back into the internal volume of the perforated pipe through the first row of holes (Figure 4).
In the initial part of the annular channel, there is an almost stagnant zone with a low intensity of heat transfer on the heat transfer surface. Figure 5 shows the distribution of the heat flux density on the working surface. The right edge in Figure 5 corresponds to the beginning of the annular channel. The red spots define the zones on the surface with the highest heat transfer, where the jet impingement of the flow occurs. Along the length of the surface blown by jets, the lowest heat flux density is observed in the initial sections of the annular gap, and the highest one in the outlet sections.

The average value of the heat transfer coefficient at the beginning of the annular gap corresponds to the natural convection level and is, on average, 11 W / m² K. To the second row, the heat transfer on the surface increases by about 15 times as a result of the jet impingement of the flow onto it. The highest heat transfer is on the half of the surface located closer to the blind end of the perforated surface. However, it should be noted that there is a significant uneven heat transfer over the working surface.

To eliminate the separation region at the inner surface of the perforated pipe, in the next design variant, the inlet end was given a smoother shape in the form of an inner quarter of a torus. The jet outflow into the annular gap occurs already from the first row of holes (Figure 6).

The considered change in the design of the flow inlet into the perforated cylinder also led to a significant increase in the average heat flow density around the perimeter (Figure 7) at the inlet section. The average heat transfer coefficient on the surface opposite the first row increased by more than 8 times compared to the initial version.
Figure 6. Distribution of the vectors of the total flow rate when the input part is in the form of an inner quarter of the torus

Figure 7. Distribution of the heat flux density when the inlet part is made in the form of an inner quarter of the torus

The most successful option for supplying flue gases to the convective stage of the recuperator is shown in Figure 8. Unlike the design shown in Figure 1, to intensify heat transfer and reduce the uneven heat transfer from flue gases to the heat transfer surface at the initial section of the convective stage, the inlet end of the perforated pipe 15 is made in the form of an inner quarter of a torus. Smooth protrusions 16 are installed on its surface, and an annular flow divider 17 is located on the inner surface of the heat transfer wall 10, which separates the front part of the annular channel 14 and forms a set of vortex chambers 18.

Figure 8. Longitudinal cross-section of the modernized recuperative-burner block

It is possible to evaluate the heat transfer intensification, due to the convective stage proposed design changes, in Figure 9 graphs. Line 1 corresponds to the change in heat transfer along the length of the convective stage for a typical design of the recuperator (Figure 4), line 2 - for the case of the inlet part in the form of an inner quarter of the torus (Figure 6). Line 3 - for the case when smooth
protrusions are additionally installed on the inlet surface, and line 4, when an annular flow divider is also located on the inner surface of the heat transfer wall. All of the above design changes lead to the intensification of heat transfer at the surface. In comparison with the typical design of the recuperative-burner unit, the heat transfer coefficient at the inlet section of the convective stage increases from 5.5 to 13 times.

The design of the recuperative-burner unit shown in Figure 8 was drawn up in the form of an application for a patent of the Russian Federation with priority certificate No. 2021106636 dated March 15, 2021.

4. Conclusions
In the initial version of the design of the recuperative-burner unit, there is a significant unevenness of heat transfer along the length and perimeter of the working surface of the convective stage.

In the initial section of the stage, a stagnant zone with the lowest heat transfer rate is observed.

To increase the flow rate of the initial section of the annular gap and enhance heat transfer on the surface in this area, the entrance to the perforated pipe should be made in the form of an inner quarter of a torus; smooth protrusions should be installed on its surface; an annular flow divider should be located on the inner surface of the heat transfer wall, which separates the front part of the annular channel with formation of a set of vortex chambers.

References
[1] Tebenkov B P 1975 *Recuperators for industrial furnaces* (Moscow: Metallurgy) p 294
[2] Saburov E, Ostashov S, Orekhov A, Leukhin Y, Nikiforov V, Kreinin E, Shaginyan I 1988 *J. Industrial energy*. 6 33–37
[3] Leukhin Y, Saburov E 2013 *Modern science: research, ideas, results, technologies* (Dnepropetrovsk: "NPVK Triakon"). 1 12 123–129
[4] Leukhin Yu L, Pankratov E V and Karpov S V 2017 *IOP Conf. Series: Journal of Physics: Conf. Series* 891 012143
[5] Leukhin Yu L, Pankratov E V 2020 *AIP Conference Proceedings* 2211 030010
[6] Dyban E, Mazur A 1982 *Convective heat transfer with jet cooling of bodies* (Kiev: NAUKOVA DUMKA) p 304
[7] Yudaev B M, Mikhailov M S and Savin V K 1977 *Heat transfer in the interaction of jets with obstacles* (Moscow: Mechanical Engineering) p 248
[8] Alekseenko S V, Dulin V M, Kozorezov Yu S, Markovich D M 2008 *Int. J. Heat and Fluid Flow* 123–128