Numerical research of the influence of the geometric parameters of shadowing fins on the intensity of jet cooling by supercritical carbon dioxide

A N Rogalev, V O Kindra, S K Osipov, B A Makhmutov and A S Zonov

National Research University "MPEI", Russia, 111250 Moscow, Krasnokazarmennaya, 14

kindra.vladimir@yandex.ru

Abstract. This paper discloses computer simulation results of thermos-hydraulic processes in cooling channels of carbon dioxide turbines used in oxy-fuel power production facilities. Stream cooling systems may be modified by the installation of shadowing ribs on the pressure and suction sides of a hollow blade and this intensifies heat transfer and reduces the cooling agent flow. The heat transfer maximal intensity is obtained at the distance between deflector orifice and shadowing fin of 0.25 orifice diameter. Change of the shadowing fin cross-section shape from oval to circular causes increases of the channel hydraulic resistance for 20–28% and the mean Nusselt number for 8–25%.

1. Introduction

The main direction of the world power industry development is the transition to environmentally friendly power sources with low toxic and greenhouse gas emissions [1, 2]. In these terms, the closed gas turbine cycles on super-critical carbon dioxide with pure oxygen combustion of organic fuels seem very perspective [3, 4].

Wide introduction of this technology requires solutions to a range of research and technical problems including the development of the equipment operating with a new heat carrier at non-traditional thermodynamic parameters. One of the crucial elements of these prospective facilities that determine production efficiency is the carbon dioxide turbine (CDT) cooling system [5, 6].

The key elements of these systems are the heat transfer intensifiers placed in the blades cooling channels, fins, pins, swirl matrixes, dimples, etc. [7, 8]. Thermo-hydraulic performance of the channels depends upon the flow swirl generator, or turbulator type and shape. The turbulator optimal choice is mostly determined by the coolant thermo-physical parameters.

The difference between the new CDT heat carrier and the traditional ones determines the importance of the cooling system modification. High specific mass and low viscosity of the carbon dioxide coolant result in high massflow amounts in the cooling channels at the dimensions of the channels retained. This factor allows the placement of additional heat transfer intensifying elements despite their higher hydraulic losses. A possible modification of the existing cooling systems is the placement of shadowing fins on the hollow airfoil walls suction and pressure sides around the deflector zone [9]. This solution increases the heat transfer by the streams deformation and reduces the high coolant flow.
It is worth mentioning that in this case, the intensifying effect is dual. Fins on the hollow airfoil walls may have a height equal to the cooling channel height. This reduces the stream's deformation by the sideways flow due to their throttling and intensifies the jet heat transfer. On the other side, the flow interacts with the wall fins, which causes the boundary layer separation under the jet, and the flow turbulence increases, and that intensifies the convective heat transfer.

With this combination of the flow turbulators, the jets eliminate stagnation zones that are usual in the fin face zones. On the other side, its shortage is the higher hydraulic resistance of the channel. This factor in carbon dioxide is not as important as in air. Thus, it is possible to conclude that this method is a prospective technology for modification of the blades cooling system. As to check this proposal and to maximize the heat transfer by the shadowing fins it is necessary to determine the fins shape and location and their influence upon the channel thermos-hydraulic performance. This paper discloses computer simulation results of the modified CDT blade cooling system.

2. Thermo-hydraulic process investigation, object and methods

2.1. Cooling channel and heat transfer intensifier performance

Figure 1 shows the simplified simulation model of the deflector installed in the blade internal cavity. The channel models had rectangular cross-section configurations that eliminate the influences of suction and pressure side curvatures.

The upper channel supplies coolant to the orifices and presents the deflector inner cavity. The lower channel simulates the space between the deflector outer surface and the cooled blade inner wall. The model main dimensions are summarized in table 1.

As to intensify heat transfer, the channel base model was modified by placement of circle and oval sections shadowing fins. The circle fins diameter was equal to the orifice diameter. The oval fins width \( L \) and length \( k \) were 0.5 and 1.0 mm. Figure 2 illustrates an example of the oval shadowing fin locations.

The dimensionless parameter \( \frac{z}{d} \) is the ratio of the distance between deflector orifice and fin \( z \) to the orifice diameter \( d \). The computer simulation involved three \( \frac{z}{d} \) values 0.15, 0.25 and 0.35.

2.2. Thermo-hydraulic process simulation

The heat transfer problem was solved in a joint approach, besides the main flow, the simulation included the channel wall analysis. The flow simulation hybrid non-structured mesh consists of tetrahedrons in the main flow zone and prisms in the wall zone (table 2). The wall heat transfer process is simulated with a non-structured mesh with the maximal element size 0.5 mm. In total, the mesh consisted of 5.7 million elements.
Table 1. Simulation model main dimensions.

| Parameter              | Value |
|------------------------|-------|
| Channel length $a$, mm | 32    |
| Channel width $b$, mm  | 10    |
| Channel height $H$, mm | 1     |
| Orifice diameter $d$, mm| 0.5   |
| Orifice transversal pitch $P_y$, mm | 2 |
| Orifice axial pitch $P_x$, mm | 2 |
| Deflector wall thickness $h$, mm | 0.5 |
| Upper channel height $S$, mm | 5 |

Figure 2. Oval shadowing fin location in the cooling channel.

Table 2. Min flow simulation mesh.

| Parameter                      | Value      |
|--------------------------------|------------|
| Element maximal size, mm       | 0.5        |
| Number of prism layers, psc    | 10         |
| First prism layer height, mm   | 0.0025     |
| The ratio of neighboring prism layers heights | 1.35 |
| Total prism layers height, mm  | 0.1916     |
| Growth law                     | wb-exponential |

The boundary conditions for simulation of the processes in cooling channels are similar to the CDT oxy-fuel cycle operating conditions [3]. Thermo-physical parameters of the carbon dioxide heat carrier are calculated with the NIST REFPROP code. At the channel inlet, the heat carrier total pressure $P_0=30$ MPa and total temperature $T_0=200°C$ were assumed. At the channel exit, the heat carrier massflow $G_0$ was tuned as to keep the constant Reynolds number 15000 at the channel inlet. The channel outer wall temperature is assumed as $T_w=850°C$.

The sideway flow influence is evaluated with the dimensionless specific side flow parameter $G_{\text{side}}/G_0$, where $G_{\text{side}}$ is the side flow and $G_0$ is the coolant flow. The following values of the specific side flow were taken: 0, 0.1, 0.5, 1.0. The simulation involved the $k$–$\omega$ turbulence model that is widely used for the cooling channel process simulation [10, 11].

2.3. Thermo-hydraulic performance analysis

Analysis of the thermos-hydraulic processes involves the dimensionless Nusselt criteria and hydraulic coefficient. The method for calculation of these parameters local and mean values is the following.

The beginning step is the calculation of the local values for thermos-hydraulic parameters. The channel cooled wall was split into transversal plates of 0.15 mm width.
Equation (1) defines the heat transfer to coolant for a plate \( i \).

\[
\alpha_{pl,i} = \frac{q_{pl,i}}{(T_{w,pl,i} - T_{fl,pl,i})}
\]  

(1)

where \( q_{pl,i} \) – heat flux density for a plate \( i \), W/m²; \( T_{w,pl,i} \) – temperature of plate \( i \), K; \( T_{fl,pl,i} \) – cooling flow temperature for a plate \( i \), K.

The mean cooling flow temperature for a plate \( i \) was calculated as the following:

\[
T_{fl,pl,i} = \frac{T_{infl,pl,i} + T_{outfl,pl,i}}{2}
\]  

(2)

where \( T_{infl,pl,i} \), \( T_{outfl,pl,i} \) – total temperature for a plate \( i \) inlet and exit, K.

The Nusselt number for a plate \( i \), or the local Nusselt number value is determined by the following equation:

\[
Nu_{pl,i} = \frac{\alpha_{pl,i} d}{\lambda_{pl,i}}
\]  

(3)

where \( \lambda_{pl,i} \) – heat conductivity of cooling agent for a plate \( i \), W/m K; \( d \) – specific hydraulic diameter of the channel section determined in the channel exit, m.

The mean heat conductivity of cooling agent for a plate \( i \) is calculated as the following:

\[
\lambda_{pl,i} = \frac{\lambda_{infl,pl,i} + \lambda_{outfl,pl,i}}{2}
\]  

(4)

where \( \lambda_{infl,pl,i} \), \( \lambda_{outfl,pl,i} \) – mean heat conductivity of cooling agent for a plate \( i \) inlet and exit, W/m K.

The mean Nusselt value in the channel was determined as the sum of its local values divided by the number of plates.

The linear hydraulic resistance coefficient in the channel is defined as the following:

\[
f_{ave} = \frac{2\Delta p d}{\rho U_{in}^2 l}
\]  

(5)

where \( \Delta p \) – channel pressure losses, Pa; \( \rho \) – coolant mean specific mass at the channel exit, kg/m³; \( U_{in}^2 \) – coolant mean flow velocity at the channel inlet, m/s; \( l \) – channel length, m.

3. Thermo-hydraulic process analysis results

3.1. Influence of side flow and cylindrical shadowing fins location

The influence of the side flow and cylindrical shadowing fins location was studied in the base turbulator shape.
Computer simulation results show that for the considered range of side flow the maximal Nusselt increase occurs at \( z/d = 0.25 \). For the case of the side flow absence, the Nusselt number at \( z/d = 0.25 \) is higher for:

- 9.2\% than the mean Nusselt value at \( z/d = 0.15 \);
- 10.6\% than the mean Nusselt value at \( z/d = 0.35 \);
- 5.6\% than the mean Nusselt value without shadowing fins.

On the other side when the specific side flow grows up to 1.0 the Nusselt number at \( z/d = 0.25 \) is higher for:

- 31.4\% than the mean Nusselt value at \( z/d = 0.15 \);
- 36.9\% than the mean Nusselt value at \( z/d = 0.35 \);
- 54.9\% than the mean Nusselt value without shadowing fins.

The heat flow density distributions in figures 3 and 4 confirm the maximal stream cooling intensity at \( z/d = 0.25 \).

![Figure 3](image_url)  
**Figure 3.** Distribution of heat flow intensity at \( G_{\text{side}}/G_0 = 0 \).

![Figure 4](image_url)  
**Figure 4.** Distribution of heat flow intensity at \( G_{\text{side}}/G_0 = 1.0 \).

It is worth mentioning that the influence of specific distance \( z/d \) upon the mean linear hydraulic resistance coefficient is small. The maximal linear hydraulic resistance coefficient \( f_{\text{ave}} \) increase occurs at \( z/d = 0.25 \). When the side flow is absent this increase is 6.4\% and when the side flow and the cooling agent flow are equal it is 13.4\%.

So, the installation of shadowing fins produces a maximal increase of the jet cooling intensity at high side flow amount, but even when there is no side flow the correct fins location remarkably increases Nusselt numbers. The study results show the optimal value of \( z/d = 0.25 \) that produces the maximal heat release efficiency.
3.2. Influence of shadowing fins configuration

For the parametric study of shadowing fins shape the specific distance between fin and orifice was assumed as optimal 0.25. The main calculation result in figures 5 and 6 shows that the cylindrical fin provides maximal stream cooling intensity. When there is no side flow the mean Nusselt number with cylindrical fins is 7.9% higher than with the oval one. When the side flow grows up to 1.0, the difference of the Nusselt number grows up to 25.3%.

On the other side, the mean hydraulic resistance with cylindrical fins is 27.9% higher than with the oval ones when there is no side flow and when the side flow grows up to 1.0 this difference is 20.7%. This effect may be explained as the following. Upstream the cylindrical fins there are low velocity zones that increase the channel hydraulic resistance and the oval shape of fins blocks these low velocity zones. Also, the velocity distribution non-uniformity with oval fins is smaller than with the cylindrical ones. Later flow separation and modified flow turbulence provide lower hydraulic resistance with the oval fins.

Thus the maximal Nusselt number values may be obtained with cylindrical shadowing fins.

Figure 5. Influence of shadowing fin shape upon local Nusselt values at \( G_{\text{side}}/G_0=0 \).

Figure 6. Influence of shadowing fin shape upon local Nusselt values at \( G_{\text{side}}/G_0=1 \).
4. Conclusions
1. Thermo-hydraulic performance of carbon dioxide turbine cooling channels with prospective flow turbulators is investigated with the Ansys CFX software.
2. Investigation of cylindrical fins location against stream cooling orifices shows that the maximal heat transfer intensity may be obtained at the distance between deflector orifice and shadowing fin 0.25 of the orifice diameter.
3. The transition from the oval shape fin cross-section to the cylindrical one produces an increase of the channel hydraulic resistance for 20–28% and the Nusselt number increase of 8–25%.

Acknowledgments
This study conducted by Moscow Power Engineering Institute was financially supported by the Ministry of Science and Higher Education of the Russian Federation (project No. FSWF-2020-0020).

5. References
[1] Matthews H D and Caldeira K 2008 Stabilizing climate requires near-zero emissions Geophys. Res. Lett. 35 L04705
[2] Davis S J et al 2018 Net-zero emissions energy systems Science 360(6396) eaas9793
[3] Rogalev A, Grigorev E, Kindra V and Rogalev N 2019 Thermodynamic optimization and equipment development for a high efficient fossil fuel power plant with zero emissions J. Clean. Prod. 236 117592
[4] Rogalev A, Kindra V, Osipov S and Rogalev N 2019 Thermodynamic analysis of the net power oxy-combustion cycle Proc. 13th European Conf. on Turbomachinery Fluid Dynamics & Thermodynamics (Lausanne, Switzerland / European Turbomachinery Society)
[5] Bolland O and Sæther S 1992 New concepts for natural gas fired power plants which simplify the recovery of carbon dioxide Energ. Convers. Manage. 33(5–8) 467–75
[6] Mathieu P and Nihart R 1999 Zero-emission MATIANT cycle J. Eng. Gas Turb. Power 121 1116–20
[7] Han J-C, Dutta S and Ekkad S 2012 Gas turbine heat transfer and cooling technology (Boca Raton: CRC Press) p 887
[8] Kindra V, Osipov S, Kharlamova D and Shevchenko I 2019 An experimental and numerical study of flow and heat transfer in cooling channels with pin fin-dimple and pin fin-groove arrays Proc. 13th European Conf. on Turbomachinery Fluid Dynamics & Thermodynamics (Lausanne, Switzerland / European Turbomachinery Society)
[9] Shevchenko I, Rogalev N, Rogalev A, Vegera A and Bychkov N 2017 Gas turbine guide vane cooled blade RU Patent 2663966 C1 13/08/2018
[10] Kim J, Bak J, Kang Y, Cho L and Cho J 2014 A study on the numerical analysis methodology for thermal and flow characteristics of high pressure turbine in aircraft gas turbine engine Korean Soc. Fluid Mach. 17(3) 46–51
[11] Rogalev A, Rogalev N, Kindra V, Osipov S and Zonov A 2019 Numerical study of heat transfer in rectangular channels with single pin fin and pin fin-dimple E3S Web Conf. 124 01010