NUMERICAL STUDY ON THE INTENSIFIED VORTEX COOLING FOR GAS TURBINE BLADE

A. Filali[1,2*], M. Boudeffa[3], F. Dous[3], Z. Nemouchi[3], O. Matar[2]

[1]Laboratoire Mécanique et système énergétique avancé, Ecole Nationale Polytechnique de Constantine, BP 75, A, Nouvelle Ville Ali Mendjli, 25000, Constantine, Algeria.
[2]Chemical Engineering Department, Imperial College London, South Kensington Campus, London SW7 2AZ, United Kingdom
[3]LEAP, Mechanical Engineering Department, Université Frères Mentouri, Constantine 1, Algeria

In the present work, numerical simulations are carried out to improve the cooling of a gas turbine blade leading edge. Various configurations having different numbers of nozzles installed between the cooling chamber and the targeted vortex chamber are investigated. Numerical computations are performed based on the k-ω turbulence model. The results show that for the same Reynolds number, *Re* = 18500, the reduction in the number of injector nozzles from 6 to 3 results in an increase in the Nusselt number of about 23.7%, but the pressure drop is also increased, and the thermal performance is decreased. Other geometry modifications are proposed that yield an increase of up to 4.12% in Nusselt number with a non-negligible increase in thermal performance.

Keywords: Gas turbine, vortex cooling, number of nozzles, cooling chamber, CFD

1. Introduction

Gas turbines are widely used in various industrial applications, particularly in the aero-propulsion of aircraft engines. To improve the power output and thermal performance of modern gas turbines, a working fluid gas with a higher temperature at the inlet of the gas turbine engine is required. Turbine blades are exposed to high-temperature stresses that can result in their deformation. As the blade's leading edge is the most critical area subjected to direct hot gases, it is necessary to design an adequate cooling mechanism in this region [1]. A series of cooling methods such as film cooling, impact cooling, and vortex cooling have been used in the leading edge of the turbine, among which vortex cooling is a new and effective method with a high heat transfer intensity and low flow losses. Several authors have reported various studies on the mechanism and characteristic parameters affecting the cooling process. Some studies focused on the influence of film holes on flow behaviour and heat transfer, e.g., Cao et al. [2], Zhou et al. [3], Yu et al. [4], and Jeong and Park [5]. Other works investigated the heat transfer and flow performance of an improved vortex in a cooling configuration. Fan et al. [6] conducted experiments and numerical simulations of a semi-cylindrical vortex chamber having five nozzles. Their results showed that higher values of Nusselt number are obtained under each nozzle for all investigated cases. However, air from downstream nozzles is easily impacted due to the "anti-cross flow" ability. Fan et al. [7] compared the behaviour of five different cooling methods, namely, impact cooling (IC) with circular nozzles, impact cooling (IC) with rectangular nozzles, vortex cooling (VC), medium-double vortex cooling (M-DVC), and tangential-double vortex cooling (T-DVC). Results showed that VC cooling had the highest heat transfer intensity, the most uniform Nusselt number distribution, and the greatest thermal performance factor, but the pressure loss in VC was much higher compared to other configurations.

In the present study, the effects of the number of nozzles, and other proposed geometrical modifications on the cooling performance of the targeted vortex chamber are investigated numerically.

2. MATHEMATICAL MODEL DESCRIPTION

In turbomachines, the equations used to solve flows are generally derived from the conservation of mass (continuity), momentum (Navier-Stokes), and energy equations.

$$\frac{\partial}{\partial x_j}(\rho u_j) = 0 \qquad (1)$$

*Corresponding Author: f.abdelkader@imperial.ac.uk

$$\frac{\partial}{\partial x_j}(\rho u_i u_j + p\delta_{ij}) = \frac{\partial \tau_{ij}}{\partial x_j} \tag{2}$$

$$\frac{\partial}{\partial x_j}\left(\rho E u_j + u_j p\right) = \frac{\partial u_i \tau_{ij}}{\partial x_j} - \frac{\partial q_j}{\partial x_j} \tag{3}$$

with u_i are the velocity vectors, E is the total mass energy, p is the static pressure, q_j is the heat flux, δ_{ij} is the Kronecker symbol and τ_{ij} are the viscous stress tensors. To these conservation equations and for an ideal gas, an equation of state adjusting the density to the thermodynamic variables is added and is defined as follows:

$$p = \rho .\ R_s .\ T \tag{4}$$

Where T is the fluid temperature, is the fluid density and R_s is the constant of perfect gas. For air R_s, air = 287 $J.kg^{-1}.K^{-1}$.

2.1. Turbulence model

The selected turbulence model for the present study is the k-ω model. This model can be written in the general form of equations 5 and 6. Transport equations of k and ω for the model are given below:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}\left[\rho u_j k - \left(\mu + \frac{\mu_t}{\sigma_k}\right)\right] = P - \beta^* \rho \omega k \tag{5}$$

$$\frac{\partial}{\partial t}(\rho \omega) + \frac{\partial}{\partial x_j}\left[\rho u_j \omega - \left(\mu + \frac{\mu_t}{\sigma_\omega}\right)\right] = \alpha \frac{\omega}{k} P - \beta \rho \omega^2 \tag{6}$$

where α*= 1, α= $\frac{5}{9}$,$\beta^* = \frac{9}{100}$, $\beta = \frac{3}{40}$,σ_k = 2, σ_ω =2, $\mu_t = \alpha^* \frac{\rho k}{\omega}$ and $P = \tau_{ij}{}^{turb} \frac{\partial u_i}{\partial x_j}$

The convergence criterion is fixed for variables to 10^{-5} except for the energy equation a value of 10^{-6} was considered to ensure a very good precision of the numerical solution.

3. Geometry, meshing strategy, and boundary conditions

The chosen geometry (reference design) selected for validation was the subject of an experimental study carried out by Fan et al. [6] and a numerical study carried by Fan et al. [7]. The geometry with the generated mesh is shown in figure 1. The mass flow rate is constant with a corresponding value of the Reynolds number equal to 18500. The total inlet temperature is 350K. The outlet mean static pressure is 0.16MPa. The target wall temperature is 500K. All other walls are considered adiabatic, and the no-slip condition is applied at all walls.

To study the dependence of the solution on the grid size, four different meshes consisting of 2.7, 3.2, 3.7, and 4.2 million nodes were considered. The relative error between the values of the average Nusselt number Nu_a obtained using 3.7 and 4.2 million nodes was 0.03%. Therefore, the mesh of 3.7 million cells was chosen for all the simulations presented in this work.

The accuracy of the heat transfer calculation is closely related to turbulence models. In terms of vortex cooling, we conducted numerical simulations to validate the present result based on the k-ω turbulence model with the experiments by Fan et al. [6] and the numerical results of Fan et al. [7] for different turbulence models and for Re = 28537.

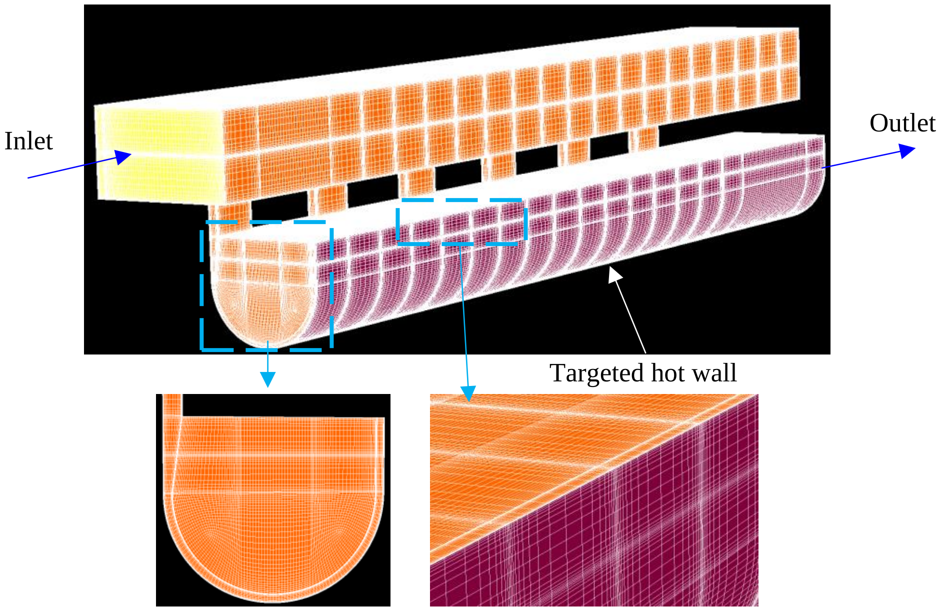


Figure 1: Meshing of the reference geometry considered in the present study proposed by Fan et al. [7].

Figure 2 shows the validation of the circumferentially averaged "Nuc" Nusselt number along the axial direction of the vortex chamber with 5 nozzles. It is shown that the general values of the different turbulence models differ considerably. The standard k-ε and k-ε RNG models have the Nusselt number that is three times larger than the experimental one, while the k-ω model has a similar value with experimental data. The k-ω model is found to be the best for modeling vortex cooling turbulence. So, the same k-ω turbulence model has been applied in the present work. As shown in figure 2, the Nuc distribution of this work is in good agreement with that of the experimental and numerical data (for the case of the k-ω model). All three have the same Nusselt number distribution along the axial direction. This indicates that the present work is well validated.

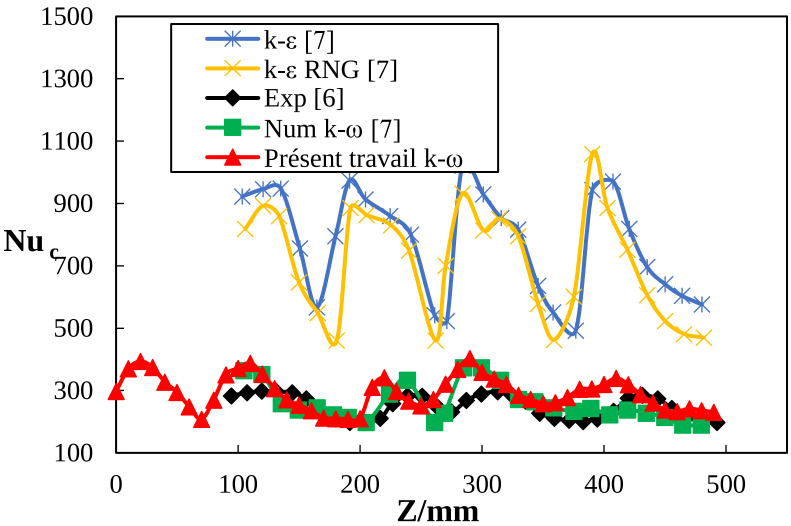
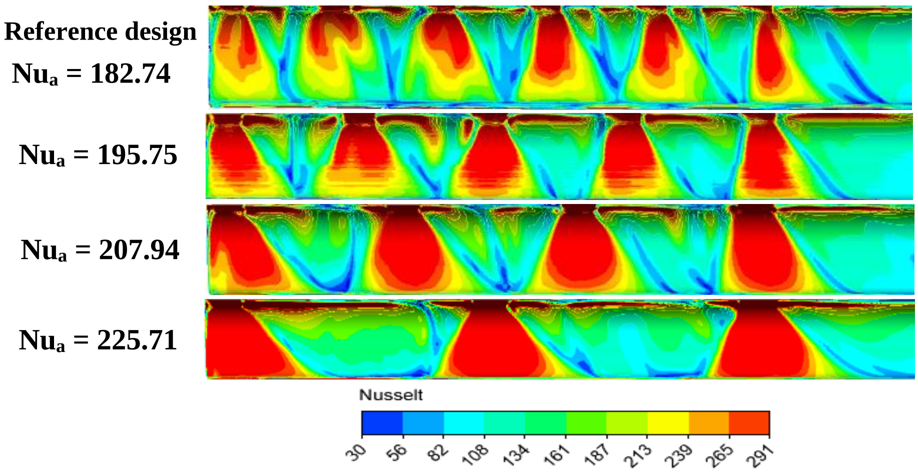


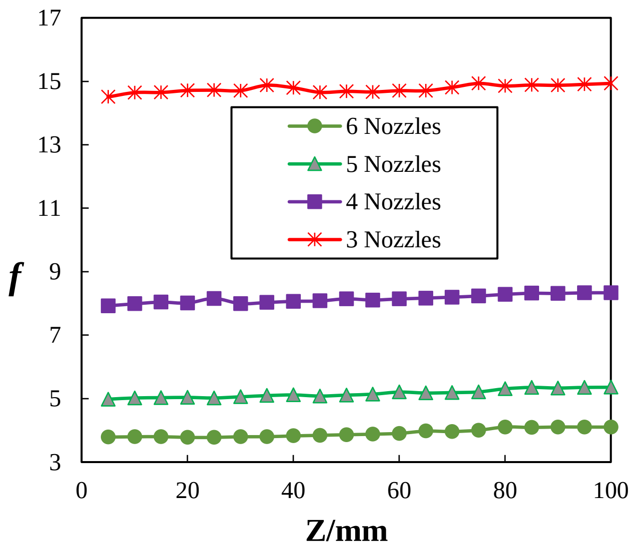
Figure 2: Validation of the present numerical result based on the k-ω turbulence model with the experimental Fan et al. [6] and the numerical results of Fan et al. [7] for different turbulence models and for Re = 28537.

4. Results and discussion

Figure 3 shows the contours of the local Nusselt number along the target wall with the computed average Nusselt number and the coefficient of friction *f* using different nozzle numbers. Generally, high values of the Nusselt number are found to exist downstream of the nozzles. This indicates good cooling in these regions. Besides, the width of the raised-Nu zones gradually increases as the number of nozzles is decreased, which is mainly due to the increased velocity through the injecting nozzles. In terms of cooling efficiency, the increase in the average Nusselt number between the reference configuration with 6 nozzles and to that with 3 nozzles is approximately 23.7% for the same Re = 18,500. In contrast, results show that decreasing the number of nozzles increases the friction coefficient, particularly in the vicinity of the nozzles exit). Therefore, the thermal performance factor defined as TPF = $\left({}^{Nua}/_{Nu_0}\right)/\left({}^{f}/_{f_0}\right)$ [7], decreases from TPF = 0.415 for the case of 6 nozzles to 0.335 for the case of 3 nozzles. Here Nu_0 and f_0 are the Nusselt number and the friction coefficient for the case of a turbulent flow in a pipe.



(a)



(b)

Figure 3: Effect of the number of the nozzles on (a) local and average Nusselt numbers, (b) friction coefficient, for a Reynolds number Re = 18,500.

Figure 4 shows three other proposed designs. In design 2, a curved plate is added downstream of each nozzle, close and parallel to the target wall. These solid surfaces serve to guide and distribute the fresh air stream along the hot wall. In design 3 the cooling chamber shape is modified from rectangular to triangular, which results in a flow acceleration through the converging passage and an intensification of the vortex in the targeted chamber. Finally, design 4 combines both modifications, i.e., curved guiding surfaces and a modified cooling chamber to a triangular shape. These modifications yielded an enhanced heat transfer compared to the original design, as presented in Figure 5. The average Nu number increase is from 182.74 for the reference design to 190.26 for design 4, which is about 4.12%. The thermal performance factor TPF increased by a small amount from 0.415 for the initial configuration to 0.42 for design 4. The use of the curved surfaces also allowed for a less inhomogeneous distribution of Nu number.

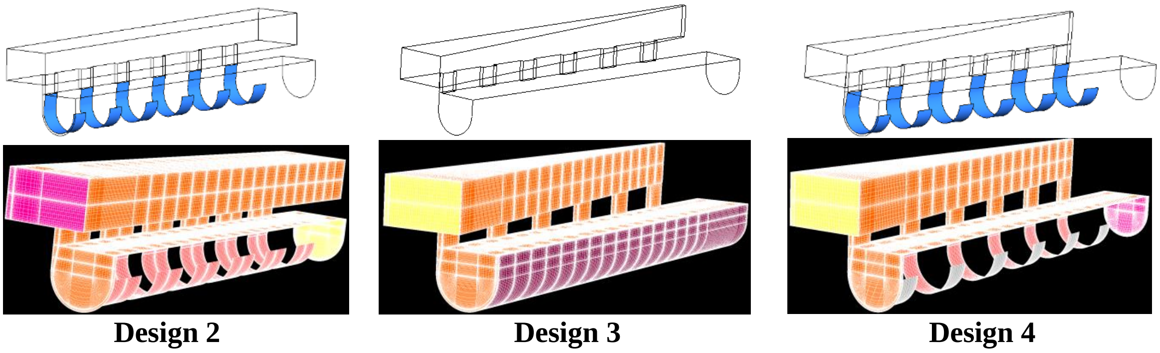


Figure 4: Geometry and meshing of the new proposed configurations

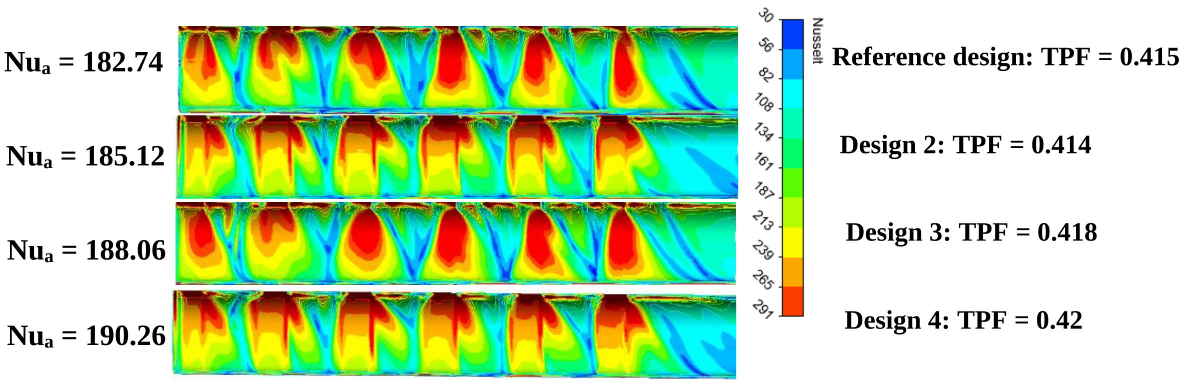


Figure 5: Local and average Nusselt numbers for the reference and new proposed designs.

CONCLUSION

It is concluded that decreasing the number of injector nozzles from 6 to 3 provides an enhanced Nusselt number of about 23.7% for the same Reynolds number Re = 18,500, but also generates an increased pressure drop. The new proposed designs are promising solutions to enhance the cooling performance, and future investigations are required to optimize these solutions.

ACKNOWLEDGEMENTS

Research Chair for OKM, Department of Chemical Engineering, Imperial College London, South Kensington Campus, London SW7 2AZ, UK.

REFERENCES

[1] J.,Yao, P., Su, J., He, J.,Wu, J., Lei, & Y. Fang, Experimental and numerical investigations on double-jet film-cooling with different mainstream incidence angles. *Applied Thermal Engineering,***166** (2020) 114737

[2] N., Cao, X., Li, Z., Wu, & X., Luo, Effect of film hole geometry and blowing ratio on film cooling performance. School of Energy and Power Engineering, *Applied Thermal Engineering.,***165** (2020) 114578.

[3] J., Zhou, X., Wang, J., Li, & H. Lu, Effects of diameter ratio and inclination angle on flow and heat transfer characteristics of sister holes film cooling. Institute of Turbomachinery, *International communication in heat and mass transfer.,* **110** (2020) 104426.

[4] Z., Yu, C., Li, B., An, J., Liu, & G., Xu, Experimental investigation of film cooling effectiveness on a gas turbine blade pressure surface with diffusion slot holes, *Applied Thermal Engineering.*,**168** (2020) 114851.
[5] Y.S., Jeong, & J.S., Park, Effect of Inlet Compound Angle of Backward Injection Film Cooling Hole. *Energies*, **13** (2020) 808.
[6] X., Fan, L., Li, J., Zou, J., Wang, & F., Wu, Local heat transfer of vortex cooling with multiple tangential nozzles in a gas turbine blade leading edge cooling passage. *International Journal of Heat and Mass Transfer, Part B*, **126** (2018) 377-389.
[7] X., Fan, L., Li, J., Zou, & k., Zhou, Cooling methods for gas turbine blade leading edge: Comparative study on impingement cooling, vortex cooling and double vortex cooling. *International communications in heat and mass transfer*, **100** (2019) 133-145.