Cooling efficiency of a NACA4412 airfoil: numerical application

Eficiência de arrefecimento de um aerofólio NACA4412: aplicação numérica

DOI: 10.54021/seesv5n1-048

Received: 01/03/2024
Accepted for publication: 22/03/2024

Toufik Tayeb Naas
PhD in Mechanical Engineering
Institution: Laboratory of System Applications of Renewable Energy, Department of Electrical Engineering, University of Djelfa
Address: Djelfa, Algeria
E-mail: t.naas@univ-djelfa.dz

Mostefa Telha
PhD in Mechanical Engineering
Institution: Mechanical Engineering Department, Faculty of Science and Technology, University of Djelfa
Address: BP 3117, Djelfa, Algeria
E-mail: mostefa.telha@univ-djelfa.dz

Ismail Ghibech
PhD Student in Agronomic Sciences
Institution: Department of Agronomic Sciences, University of Biskra, Hydraulics Department, Faculty of Science and Technology, University of Djelfa
Address: BP 145, RP 07000, Biskra, Algeria, BP 3117, Djelfa, Algeria
E-mail: ismail.ghibech@univ-biskra.dz

Omar Mokhtar Khelifa
Master in Mechanical Engineering
Institution: Mechanical Engineering Department, Faculty of Science and Technology, University of Djelfa
Address: BP 3117, Djelfa, Algeria
E-mail: omarkhelifa777@gmail.com

Salem Ben Abdelhafid
Master in Mechanical Engineering
Institution: Mechanical Engineering Department, Faculty of Science and Technology, University of Djelfa
Address: BP 3117, Djelfa, Algeria
E-mail: abedhafid@gmail.com

ABSTRACT
The temperatures imposed on the blades of the first stages of turbines are generally very high; these expose the latter to harmful thermal effects, pushing
manufacturers to continually improve techniques for cooling the blades. It is true that by increasing the temperature of the gases at the inlet of the turbines, we increase the efficiency, the performance of the machines, and we improve the power and fuel consumption with a significant reduction in polluting gases. Thus, the current general trend among manufacturers is to design machines that operate at increasingly high inlet temperatures. This has led, therefore, to the constant search for new materials with high thermal resistance and to constantly improve cooling techniques. This task is conditioned by a good and deep understanding of the phenomenon of heat transfer in turbine blades. This study examines the three dimension numerical simulation of the flow and heat exchange inside an internal cooling channel of a gas turbine blade with the profile of NACA 4412. This channel plays an important role in increasing heat exchange between the cooling air and the walls of the blade. In the turbulent regime, we have investigated the cooling of a profile blade NACA 4412 using forced convection (V = 200, 250, 300, 350, and 400 m / s). determined that the value of the cold air's speed increases with the intensity of the secondary flow inside the morning. Results found that, The flow dynamics and kinematics of fluid particles alter significantly when the cooling air speed is increased, and the cooling level is enhanced.

**Keywords:** cooling, efficiency, airfoil, CFD, NACA4412.

**RESUMO**
As temperaturas impostas às pás dos primeiros estágios das turbinas são geralmente muito altas; elas expõem as pás a efeitos térmicos prejudiciais, levando os fabricantes a aprimorar continuamente as técnicas de resfriamento das pás. É verdade que, ao aumentar a temperatura dos gases na entrada das turbinas, aumentamos a eficiência, o desempenho das máquinas e melhoramos a potência e o consumo de combustível com uma redução significativa dos gases poluentes. Assim, a tendência geral atual entre os fabricantes é projetar máquinas que operem com temperaturas de entrada cada vez mais altas. Isso tem levado, portanto, à busca constante de novos materiais com alta resistência térmica e ao aprimoramento constante das técnicas de resfriamento. Essa tarefa está condicionada a um bom e profundo entendimento do fenômeno da transferência de calor nas pás das turbinas. Este estudo examina a simulação numérica em três dimensões do fluxo e da troca de calor dentro de um canal de resfriamento interno de uma pág de turbina a gás com o perfil NACA 4412. Esse canal desempenha um papel importante no aumento da troca de calor entre o ar de resfriamento e as paredes da pá. No regime turbulento, investigamos o resfriamento de uma pág de perfil NACA 4412 usando convecção forçada (V = 200, 250, 300, 350 e 400 m/s). Determinamos que o valor da velocidade do ar frio aumenta com a intensidade do fluxo secundário dentro da manhã. Os resultados revelaram que a dinâmica do fluxo.

**Palavras-chave:** arrefecimento, eficiência, aerofólio, CFD, NACA4412.
1 INTRODUCTION

The temperatures imposed on the blades of turbines are generally very high; these expose the latter to harmful thermal effects, pushing manufacturers to continually improve techniques for cooling the blades. It is true that by increasing the temperature of the gases at the inlet of the turbines, we increase the efficiency, the performance of the machines, and we improve the power and fuel consumption with a significant reduction in polluting gases.

Thus, the current general trend among manufacturers is to design machines that operate at increasingly high inlet temperatures. Therefore, the constant search for new materials with high thermal resistance and to constantly improve cooling techniques. This task is conditioned by a good and deep understanding of the phenomenon of heat transfer in turbine blades.

Obviously, air is the cooling fluid usually used both in the field of aviation and in the industrial field. The cooling air is generally drawn off at the outlet of the compressor, introducing either a decrease in the efficiency of the machine or an increase in fuel consumption. Engineers are thus required to optimize the quantity of air extracted so that the overall machine performance would be only slightly affected, while gaining protection of the blades through cooling.

Many investigations were conducted on flat plates to obtain the basic film cooling characteristics downstream of a single hole or one row of hole with only stream wise angle [1-5].

Zhang et al investigated an aerothermal characteristics of a blade tip with the tangential jet cooling scheme using the Pressure Sensitive Paint (PSP) technique under transonic flow conditions. Their experiment was carried out at tip clearance gaps of 0.7% and 1.5% and four mass flow ratios. The inlet Reynolds number and exit Mach number are 370 000 and 1.05, respectively. They found that the secondary flow covered .The jet cooling scheme of the bottom wall of the tip cavity. For the mass flow ratio increasing, the cooling effectiveness (η) of the whole blade was improved [6].

Zhiyu et al were carried out to study the effects of compound angle, hole arrangement, and blowing ratio on the film cooling performance of multiple rows of holes on the suction surface of a turbine blade. The turbine worked at rotational speed of 600 rpm corresponding to the rotational Reynolds number of 5.36x10⁵.
Three rows of cylindrical holes arranged in line or in stagger were drilled on the rotor blade suction surface at the streamwise location of 12.4%, 17.8%, and 23.2%, respectively. Three compound angles, with the same streamwise angle of 45 but different lateral deflection angles of 45, 0, and 45, were studied. The film cooling effectiveness was obtained using pressure sensitive paint (PSP) technique with average blowing ratios varied from 0.5 to 2.0. The results showed that the application of compound angle changes the jet direction in the near-hole region and makes the film spread laterally. Compared with the film cooling without compound angle, using positive and negative compound angle can improve overall average film cooling effectiveness by about 20%.

Havenetal. [7] studied the development of CRVP in detail. The results showed that CRVP is developed from the vortex pair existed in the hole side wall boundary layer. The effect of CRVP can be weakened by changing geometry.

Many studies on the film cooling of shaped hole [8, 9] and combined hole system [10,11] were carried out. However, both of the hole geometries have certain requirements on the processing technology Ahn et al. [12] studied the film cooling performance on the leading edge of a rotor blade under rotating condition. Li et al. [13] studied the film cooling performance on both Suction side and pressure side of a rotor blade. Due to the effect of rotation, the deflection of the film trajectories is slightly enhanced as the rotational speed increases. Besides, the main stream Reynolds number increases with the increasing rotational speed which improves the cooling performance at a small blowing ratio [14].

Ahmed et al studied numerically the heat transfer analysis of nickel-chromium and titanium carbide gas turbine blades cooling. The heat transmission of gas turbine blades using two materials and 3 cooling methods is investigated. The cooling of blades has been suggested using a variety of approaches such as internal cooling and film cooling. A turbine blade was designed using CATIAV5-6R. Nickel Chromium, Titanium carbide materials were considered for the study. The CFD software FLUENT is used to analyze the heat transport of a gas turbine blade. Analysis was carried out for internal cooling and film cooling. Titanium carbide has been shown to have better thermal properties than Nickel-Chromium.

Zhang et al experimental investigated the film cooling performance of a turbine blade tip with a trapezoidal slot cooling scheme in transonic flow using PSP
technique. A turbine blade tip with a trapezoidal slot cooling scheme is first proposed in the current paper. The film cooling characteristics of a turbine blade tip with a trapezoidal slot scheme are experimentally studied by the pressure sensitive paint (PSP) technique in transonic flow. Varieties of density ratio, tip clearance gap and position of the trapezoidal slot scheme are selected as investigation parameters. The mainstream Reynolds number is 370,000 based on the blade axial chord length. The cascade exit Mach numbers are is 1.05. The results indicate that increasing the density ratio has a positive effect on the film cooling effectiveness of the whole blade tip with a pressure side trapezoidal slot scheme at a small tip clearance gap, whereas it decreases the film cooling effectiveness upstream of the blade tip at the large tip clearance gap. For the blade tip with a pressure side trapezoidal slot scheme, the film cooling effectiveness of the tip clearance gap of 1.5% near the trailing edge is higher than that of the other tip clearance gaps at the same density ratio condition. Changing the trapezoidal slot position from the tip pressure side to the tip midline increases the film cooling effectiveness near the trailing edge.

Zhao conducted the the flow trends of gas mainstream and coolant ejection in the blade fluid domain are closely related to the non-uniform profile characteristics of gases before the turbine stage. The validity of the non-uniform pressure and temperature profile for the mainstream was determined under the typical working conditions of a torsion blade by comparing the differences between the experimental and numerical simulation results of the leading-edge Mach numbers for various mainstream cases. Then, the film cooling effectiveness of the showerhead holes at the leading edge of the gas turbine twisted blade under the non-uniform mainstream was analyzed. The influence mechanism and rule of the radial angle and stream-wise torsion angle on the cooling effectiveness of the leading-edge showerhead were analyzed using the jet characteristics of single film and the span-wise average film cooling effectiveness, respectively. Finally, analyzing the influence of different coupling angles on the cooling effectiveness of the leading-edge region provided the optimal angle arrangement scheme. The results show that the reasonable coupling angle arrangement of showerhead holes can enhance the cooling effectiveness in the leading-edge region. Thus, the
coupling angle arrangement should be considered in the engineering design for the leading edge of gas turbine blades.

Xiaojun [15] explored a new cooling method to further improve the cooling performance for the gas turbine blade leading edge, a vortex double wall cooling configuration is established and studied. The numerical simulation was conducted after grid independence validation and turbulence model validation. Four groups of different combined cooling methods, different disturbing objects, different rows of bridge holes and the existence of film holes were studied in detail. Results showed that the vortex double wall cooling method could improve the cooling performance by more than 3 times than the basic straight passage double wall cooling. And compared to impingement double wall cooling, the vortex double wall cooling has better cooling behavior with larger inner surface Nusselt number and globally-averaged Nusselt number. Different disturbing objects on the outer surface could improve the local heat transfer performance without affecting the overall cooling performance, and protrusions are found to be a good kind of disturbing objects. When rows of bridge holes increase to 3, its thermal performance factor is 23.6% higher than case with 1 bridge hole row. It is regarded that increasing bridge hole rows could bring considerable cooling improvements. For the condition with film holes, it is found the overall flow and heat transfer performance is 6.2% higher than case without film holes. Generally, the vortex double wall cooling method proves to be a potential cooling method for the leading edge, and the configuration with protrusions and more rows of bridge holes could further improve its cooling behavior.

Jet impingement is applied mainly for the cooling of the thermally high loaded leading deration. Multiple jets are, also, observed to be utilized in double wall configurations, which presents further increasing cooling performances for the turbine internal flow [16-18].

Flow and heat transfer characteristics of the jet impingement heat transfer with non-rotating conditions have been intensively explored in previous reports. From the existing literature, the heat transfer of the impingement system depends on the following factors: jet and feeding channel configurations, Reynolds number, and direction of cross flow removal.
Elston and Wright et al [19] conducted an experimental investigation on heat transfer in an impinging jet row semi cylinder channel at $Re_j =$6000–24,000 and $Ro_j =0–0.13$

Meanwhile, the leading orientation deteriorate the heat transfer 66,70.

Overall, the heat transfer results by jet impingement are similar to the traditional two pass channels, but with the additional complexity of cross flow and jet-to-surface distance.

A part from that, modern studies attempted to increase heat transfer on the impingement channel by modifying the impingement surface. Roughness elements on the surface can increase turbulence on the impingement channel and guided the wall jet flow, resulting in a further increase in the heat transfer. [20].

Moreover, it was examined by several researchers that the combination of jet impingement with obstructed elements and external effusion holes reduced negative cross flow effects, resulting in higher heat transfer enhancement. [21]

Many recent investigations attempt to reduce the negative effects by modifying the dimpled shapes. [22-23].

Most of the results from the modified dimples reduce flow separation and recirculation flow inside the cavity or even generate strong secondary flows developed in the dimple, which further enhances the heat transfer. For stationary conditions, it can be safely deduced that the roughness walls showed an enhancement of the heat transfer over the smooth surface.

The objective of this study is to examine a three-dimension numerical simulation of the flow heat exchange and get the better cooling technic inside gas turbine blade.

2 PHYSICAL MODEL AND CFD MODELING

A schematic of the physical model alongside the relevant boundary conditions is shown in figure 1.
The hydraulic diameter is 5.90 mm; \( Dh = \frac{4 \cdot \text{right section of the pipe}}{\text{wet perimeter}} \)

\[ Dh = \frac{4 \cdot [(5.77 + 3.51) \cdot 8.32 / 2] / (8.32 + 8.65 + 5.77 + 3.71)}. \]

The unfolded length of the flow passage geometry is equal to 275 mm

The mass conservation, Navier–Stokes and energy equations, which given by equations (1), (2) and (3) respectively, are numerically solved by using a CFD code. In this study, the fluid is considered as incompressible, Newtonian while the flow regime is steady and laminar:

\[
\text{div}\vec{V} = 0 \tag{1}
\]

Where \( \vec{V} \) is the velocity vector.

\[
\vec{V} \cdot \nabla \vec{V} = -\frac{1}{\rho} \vec{V} \cdot \nabla P + \text{div}\tau \tag{2}
\]

Where \( \rho \) is fluid density, \( \tau \) (Pa) is the shear stress and \( P \) is the pressure.

\[
\rho c\vec{V} \cdot \nabla T = \lambda \Delta T \tag{3}
\]

The applied boundary conditions are:

at the inlet section, uniform velocity profile and the temperature equal to \( T_i = 300 \text{ k} \).

at solid walls, \( T_w = 900-1100 \text{ k} \) and no–slip conditions.

at the outlet section, the pressure outlet condition is considered.

Reynolds number defined as:
Re = \frac{\rho U D_h}{\mu} \quad (4)

Heat transfer coefficient, \( h \), for wall heat flux boundary condition is given as:

\[ h = \frac{q''}{(T_b - T_w)} \quad (5) \]

Where, \( q'' \) (w/m²) is the wall heat flux, \( T_b(k) \) is the mean bulk temperature fluid over the cross-sectional area and \( T_w(k) \) is perimeter average wall temperature.

These two temperatures are defined as:

\[ T_w(s) = \frac{1}{P} \int_P T_w \, dp \quad (6) \]

\[ T_b(s) = \frac{1}{A U_i} \iint_A \vec{V} \cdot \vec{n} T \cdot dA \quad (7) \]

The mean heat transfer coefficient, \( h_{mean} \), defined as:

\[ h_{mean} = \frac{1}{L} \int_0^L h(s) \, ds \quad (8) \]

The local Nusselt number given by the following equation:

\[ Nu_{local} = h(s) \frac{D_h}{\lambda} \quad (9) \]

Where, \( \lambda \) is the thermal conductivity of the fluid (\( \lambda = 0.6 \text{W} \cdot \text{s}^{-1} \cdot \text{K}^{-1} \)).

And the mean Nusselt number is defined by:

\[ Nu_{mean} = \frac{1}{L} \int_0^L Nu_{local} \, ds \quad (10) \]
3 MESH SENSITIVITY

Table 1 shows the static pressure, velocity, and temperature rates for the outlet section of the proposed blade turbine for all mesh densities. It shows that the differences between the two elements mesh respectively 250230 and 300598 of variation of speed, pressure and temperature of less than 0.24%, 1.4% and 1.4%. This presents another argument to select the 250 230 elements as the optimal mesh density for the rest of the calculations.

Table 1: The static pressure, velocity, and temperature rates for the outlet section

<table>
<thead>
<tr>
<th>Eléments</th>
<th>Pression (Pa)</th>
<th>Vitesse (m/s)</th>
<th>Température (K)</th>
</tr>
</thead>
<tbody>
<tr>
<td>100517</td>
<td>-1720.7978</td>
<td>374.18526</td>
<td>475.53995</td>
</tr>
<tr>
<td>150977</td>
<td>-2042.7024</td>
<td>380.42642</td>
<td>474.50165</td>
</tr>
<tr>
<td>200681</td>
<td>-2073.7298</td>
<td>381.41881</td>
<td>472.18551</td>
</tr>
<tr>
<td>300598</td>
<td>-2079.131</td>
<td>381.32889</td>
<td>473.99383</td>
</tr>
</tbody>
</table>

Source: Authors.

Figure 2: pressure profiles for different mesh densities at the outlet section, (A) X- coordinates (b) Y-coordinates.

Source: Authors.

Figure 3: Temperature profiles for different mesh densities at the outlet section, (A) X-coordinates (b) Y-coordinates.

Source: Authors.
4 RESULTS AND DISCUSSIONS

Flow and thermal performances of cooling blade system are studied in detail for NACA4412 airfoil. These performances are investigated as function of inlet velocities ranging from 200 to 400 m/s and different external heat temperature 900, 1000, 1100 and 1200 K.

The temperature contours of the outlet section of the vane for \( V = 200, 300 \) and \( 400 \) m/s, are shown in Figures 4 and 5. For a given speed value and on the same external heating, the quality of cooling is more vigorous when the generalized speed is higher.

Increasing the speed of the cooling air greatly improves the flow dynamics and kinematics of fluid particles change considerably and the cooling level will be improved. These model changes for different external temperatures and their physical implications are discussed in the next section (Figure 5) vane consisting of \( V = 200 \) m/s, where the maximum edge temperature is lower to that of the blade made up of box \( V = 400 \) m/s.

![Figure 4: Temperature contours of the middle section of the vane for input speeds V = 200, 300 and 400 m/s.](source: Authors.)
Figures 6 and 7 show the static pressure contours for various cooling air velocities at the turbine inlet temperature $T = 900$ K, 1000 K, 1100 K and 1200 K as the speed increases, for different temperatures, the agitation is more vigorous due to the chaotic kinematics of the trajectories and the existence of intense recirculation zones in the flow. This behavior contributes considerably to the improvement of cooling performance and pressure drop also increases (Figure 6). In addition, the heat transfer is more vigorous, and therefore the pressure values change faster; the best cooling quality is obtained in this type of geometry, the cooling time does not exceed one second for all values of the speed cold air and for different turbine inlet temperatures. Note that the chaotic nature of the trajectories in this blade geometry has a complete effect on the thermal cooling system taking into account the conduction of the wall.
Figure 6: Pressure contours of the outlet section of the vane for an inlet velocity \( V = 200, 300 \) and \( 400 \) m/s.

![Pressure contours of the outlet section of the vane for an inlet velocity V = 200, 300 and 400 m/s.](image)

Source: Authors.

Figure 7: Pressure contours of the fluid flow inside the vane for an inlet velocity \( V = 200, 300 \) and \( 400 \) m/s.

![Pressure contours of the fluid flow inside the vane for an inlet velocity V = 200, 300 and 400 m/s.](image)

Source: Authors.

Figures 8 and 9 show the fluid flow velocity contours of the vane turbine side for \( V = 200, 300 \) and \( 400 \) m/s, at various external heated vanes ranging from 900k to 1200k.
The intensity of the secondary flow inside the blade increases with the increase in the value of the speed of the cold air. It is evident from the figure that as the secondary flow becomes more skewed towards the wall of the dawn.

Figure 8: Contours de vitesse de la section de sortie de l’aube pour une vitesse d’entrée \( V = 200, 300 \) et \( 400 \) m/s.

<table>
<thead>
<tr>
<th>( V = 200 ) m/s</th>
<th>( V = 300 ) m/s</th>
<th>( V = 400 ) m/s</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image1.png" alt="Image" /></td>
<td><img src="image2.png" alt="Image" /></td>
<td><img src="image3.png" alt="Image" /></td>
</tr>
<tr>
<td>( T = 900 ) K</td>
<td>( T = 1000 ) K</td>
<td>( T = 1200 ) K</td>
</tr>
</tbody>
</table>

Source: Authors.

Figure 9: Velocity contours of the fluid flow inside the vane for an inlet velocity \( V = 200, 300 \) and \( 400 \) m/s.

<table>
<thead>
<tr>
<th>( V = 200 ) m/s</th>
<th>( V = 300 ) m/s</th>
<th>( V = 400 ) m/s</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image4.png" alt="Image" /></td>
<td><img src="image5.png" alt="Image" /></td>
<td><img src="image6.png" alt="Image" /></td>
</tr>
<tr>
<td>( T = 900 ) K</td>
<td>( T = 1000 ) K</td>
<td>( T = 1200 ) K</td>
</tr>
</tbody>
</table>

Source: Authors.
Figures 10, 11 and 12 show the changes in the pressure, temperature and speed rates as a function of the speeds ranging from 200 to 400 m/s at the output X-Y coordinates, with an external temperature of T = 900 K. In all cases of heated vane, as heat transfer occurs by conduction inside the vane body, cooling increases with increasing cooling air velocity.

The evaluation of heat transfer rates and velocities shows that perfect cooling is achieved quickly in the chaotic geometry. In addition, it is proved by the section rates that the temperature field is well and rapidly cooled in the chaotic flow created in the dawn flow, see Tables 2, 3, 4 and 5.

<table>
<thead>
<tr>
<th>Pressure</th>
<th>Temperature</th>
<th>Velocity</th>
</tr>
</thead>
<tbody>
<tr>
<td>T=900 K ; V=200 m/s</td>
<td>-655.95016</td>
<td>493.21001</td>
</tr>
<tr>
<td>T=900 K ; V=250 m/s</td>
<td>-1039.1276</td>
<td>485.00548</td>
</tr>
<tr>
<td>T=900 K ; V=300 m/s</td>
<td>-1512.4772</td>
<td>478.13498</td>
</tr>
<tr>
<td>T=900 K ; V=350 m/s</td>
<td>-2073.4713</td>
<td>472.18601</td>
</tr>
<tr>
<td>T=900 K ; V=400 m/s</td>
<td>-2724.573</td>
<td>466.92898</td>
</tr>
</tbody>
</table>

Source: Authors.

Table 3: Evaluation of the pressure, temperature and speed at the outlet of the turbine at T= 1000 K

<table>
<thead>
<tr>
<th>Pressure</th>
<th>Temperature</th>
<th>Velocity</th>
</tr>
</thead>
<tbody>
<tr>
<td>T=1000 K ; V=200 m/s</td>
<td>-656.24775</td>
<td>525.41226</td>
</tr>
<tr>
<td>T=1000 K ; V=250 m/s</td>
<td>-1039.1276</td>
<td>515.83965</td>
</tr>
<tr>
<td>T=1000 K ; V=300 m/s</td>
<td>-1512.4772</td>
<td>507.82406</td>
</tr>
<tr>
<td>T=1000 K ; V=350 m/s</td>
<td>-2073.4713</td>
<td>500.88361</td>
</tr>
<tr>
<td>T=1000 K ; V=400 m/s</td>
<td>-2724.573</td>
<td>494.7504</td>
</tr>
</tbody>
</table>

Source: Authors.

Table 4: Evaluation of the pressure, temperature and speed at the outlet of the turbine at T= 1100 K

<table>
<thead>
<tr>
<th>Pressure</th>
<th>Temperature</th>
<th>Velocity</th>
</tr>
</thead>
<tbody>
<tr>
<td>T=1100 K ; V=200 m/s</td>
<td>-655.95016</td>
<td>557.61321</td>
</tr>
<tr>
<td>T=1100 K ; V=250 m/s</td>
<td>-1039.1276</td>
<td>546.67383</td>
</tr>
<tr>
<td>T=1100 K ; V=300 m/s</td>
<td>-1512.4772</td>
<td>537.51315</td>
</tr>
<tr>
<td>T=1100 K ; V=350 m/s</td>
<td>-2073.4713</td>
<td>529.58121</td>
</tr>
<tr>
<td>T=1100 K ; V=400 m/s</td>
<td>-2724.573</td>
<td>522.57183</td>
</tr>
</tbody>
</table>

Source: Authors.
Table 5: Evaluation of the pressure, temperature and speed at the outlet of the turbine at $T=1200$ K

<table>
<thead>
<tr>
<th>Pressure</th>
<th>Temperature</th>
<th>Velocity</th>
</tr>
</thead>
<tbody>
<tr>
<td>$T=1200$ K; $V=200$ m/s</td>
<td>-655.95016</td>
<td>589.81481</td>
</tr>
<tr>
<td>$T=1200$ K; $V=250$ m/s</td>
<td>-1039.1276</td>
<td>577.508</td>
</tr>
<tr>
<td>$T=1200$ K; $V=300$ m/s</td>
<td>-1512.4772</td>
<td>567.202 24</td>
</tr>
<tr>
<td>$T=1200$ K; $V=350$ m/s</td>
<td>-2073.47 13</td>
<td>558.27881</td>
</tr>
<tr>
<td>$T=1200$ K; $V=400$ m/s</td>
<td>-2724.573</td>
<td>550.39326</td>
</tr>
</tbody>
</table>

Source: Authors.

Figure 10: Static pressure at the outlet of the X-Y axis for $V=200$ to 400 at the heated external vane $T=900$ k.

Source: Authors.

Figure 11: Temperature profile at the outlet of the X-Y axis for $V=200$ to 400 at the heated external vane $T=900$ k.

Source: Authors.
Figure 12: Velocity profile at the outlet of the X-Y axis for V = 200 to 400 at the heated external vane T = 900 k.

Source: Authors.

Figure 13: Static pressure at the outlet of the X-Y axis for V = 200 to 400 at the heated external vane T = 1200 k.

Source: Authors.

Figure 14: Temperature profile at the outlet of the X-Y axis for V = 200 to 400 at the heated external vane T = 1200 k.

Source: Authors.
5 CONCLUSION

The objective of our work was the numerical simulation of the cooling of the blades of a gas turbine. We have studied the cooling of a profile blade NACA 4412 with forced convection (V = 200, 250, 300, 350, and 400 m / s) in the turbulent regime. Concluded that the intensity of the secondary flow inside the dawn increases with the increase in the value of the speed of the cold air. It is evident from the figure that as the flow secondary becomes more skewed towards the dawn wall. visualization of heat transfer and path-lines presents that the chaotic configuration is inefficient at turbulent regime, while the new designs exhibit rapid mixing flow and cooling technic with lower pressure losses. Thus, it can be used to enhance the homogenization in several cooling turbine systems. As future work, incorporating conjugate heat transfer modeling to simulate the interaction between fluid flow and solid structures, leading to more accurate temperature predictions, and using multi-objective optimization techniques to simultaneously optimize cooling performance, pressure loss, and structural integrity of the components. Cooling blades play a crucial role in improving the energy efficiency of various systems, such as gas turbines, aircraft engines, and computer servers. Academic research can focus on developing more efficient cooling blade designs to reduce energy consumption and carbon emissions, which aligns with the global effort to combat climate change. Community research can explore how implementing these more efficient designs can lead to cost savings and environmental benefits for industries and households.
REFERENCES


