Incidence Angle Effect on the Behaviour of a Flow around the Turbine Blades

MEROUANE HABIB, DRAI ISMAIL, SAHNOUN RACHID
Department of Mechanical Engineering,
University of Mascara,
ALGERIA

Abstract: - Our work involves a numerical simulation of a turbulent flow that crosses a turbine blade for different orientation angles using a computational fluid dynamics (CFD) code. The turbine's performance mainly depends on the characteristics and efficiency of the blades for optimal orientation angle position. The numerical simulation is based on the finite volume method for the differential equations discretization; different schemes have been used to solve these equations such as first upwind, second upwind, and power law. The turbulence model used in this study is the standard k-epsilon model. The equations governing the flow are solved by the Semi-Implicit Method for Pressure-linked Equations SIMPLE algorithm. The results obtained allow for identifying and characterizing the flow around the blade by plotting the dynamic field, the pressure coefficient, and the Mach number.

Key-Words: - Turbine blade, Turbulent flow, blades, Turbine, Turbine flow, Turbine performance.

Received: July 22, 2024. Revised: November 11, 2024. Accepted: December 13, 2024. Published: February 10, 2025.

1 Introduction

Turbomachinery is master machines in strategic fields such as transportation (aircraft engines, automotive turbochargers, etc.) and energy production (steam turbines, gas turbines, and compressors). Due to the strong requirements of reliability, safety, flexibility of use, and respect for the environment imposed on these machines, manufacturers must have increasingly fine and precise analysis tools from the design stage. In this context, blades are therefore the subject of special attention. Turbomachinery is composed of one or more series of blades alternately on the stator and the rotor. Thermal turbomachinery allows the conversion of thermal energy into mechanical energy via a fluid. The energy transformation between the rotating shaft and the continuous and occurs through rotating blades.

The principle blading choice of a turbomachinery is to achieve the required deflection with a minimum of loss, it is also required that the blading operates without separation in a wide range outside the nominal operating point of the machine.

Several numerical researchers and experimenters have studied the effects of the blades orientation on the behavior of the flows that cross the fixed and moving elements of the turbines, [1], [2], [3], [4], [5], [6], [7], [8], [9], [10], [11], [12], [13].

The transonic and low supersonic flow losses of two nozzle blades for steam turbines are studied by [14]. They noted that the effect of Mach numbers on losses was much more significant than that due to the very large incidences of transonic and weak supersonic flows.

The effect of incidence on the aerodynamic performance of the turbine guide cascade is studied by [15]. The calculation results indicated that the blades with a backward loading profile had a good adaptation to the incidence angle. After changing the incidence angle, the incidence angle affected the static surface pressure distribution less on most of the pressure surface and suction surface span than in other locations. However, an incidence angle of -20° would decrease the pressure loss in a narrow range, and an incidence angle of 20° would increase the pressure loss.

The effect of strength on the performance of gas turbines operating under operational conditions is studied by [16]. Their objective was to evaluate the feasibility of adopting high-lift and low-strength designs for incidence-tolerant blade profiles. Their results showed that strength has both positive and negative effects on losses over a wide range of incidence angles. Increasing the pitch-to-chord ratio (the inverse of strength) not only reduces the number of blades but also improves performance in negative incidences. Their results also indicated that it is possible to use high-lift designs with optimized leading edges and maximum deflection for incidence-tolerant blade profiles.

The authors [17] studied the experiment and the numerical simulation of the aerodynamic performance of a turbine with different incidence angles. Their results showed that changing the inflow angle affects the pressure distribution inside the blade passage. During the incidence from negative to positive, the high load region moves towards the leading edge and the pressure gradient increases leading to a significant enhancement of the passage vortex. The effect of the inflow angle on the cavity interior occurs near the leading edge of the blade, while on the blade passage, it occurs after the pressure branches of the vortex impact the suction surface. Both constraints result in a small variation of the leakage loss with the inflow angle.

The effects of incidence angle in large-scale transonic turbine cascade conditions are studied by [18]. Aerodynamic measurements showing the effects of large variations in incidence angle on a set of HPT turbine blades, total pressure, and mid-span flow angle measurements were obtained at seven inlet flow angles. Their results showed that large negative incidence causes the pressure surface and suction surface loads to reverse. At a fixed negative incidence, the load pattern remains relatively constant as the Reynolds number varies at a fixed Mach number.

The authors [19] studied the film cooling performance of a leading edge with three rows of film holes on an enlarged turbine blade in a linear cascade by using the experiment. The effects of blowing ratio, inlet Reynolds number, isentropic outlet Mach number, and off-design incidence angle (i<0°) are considered. When the incidence angle decreases from −15° to 0°, the film cooling efficiency on the pressure side decreases and shows the opposite trend on the suction side. At an off-design incidence of −15° and −10°, there is a low peak following the leading edge on the pressure side caused by the separation bubble, but it disappears when the incidence and blowing ratio increase.

The effect of flow incidence on the aero-thermal performance of transonic flows with blade tip clearance is studied by [20]. Five incidences within the range of −15 to 15° were studied. Their results show that the effects of flow incidence on aerodynamics and heat transfer in the forward part of the blade tips are more significant than that in the aft part of the blade tips, and the effects of flow incidence at 0.4% tip gap are smaller than that at 1.0% tip gap. At various incidences, as the tip gap height increases, the shock wave becomes more significant.

The authors [21] studied by the experiment and the numerical simulation the effects of flow incidence angles on the performance of a cascade of stator blades of a high-pressure steam turbine. Their study aims to quantify the effects of off-design operation on the aerodynamic performance of a linear cascade of high-deflection steam turbine blades. The flow incidence on the leading edge of the blades varied from −15.3° to +21.0°. Their results showed that when the flow incidence increases in the positive direction, a limited under-expansion of the flow on the pressure side and overexpansion on the suction side of the blade in the leading edge area of the cascade channel is observed. An opposite trend is observed for the negative incidence cases. [22], studied by the experiment the aerodynamic losses of a high-pressure steam turbine nozzle subjected to a wide range of incidence angles (−34° to 26°) and different exit Mach numbers. They observed that the total pressure loss coefficient increased by a factor of 7 compared to the total pressure losses measured at subsonic conditions. For the tested incidence range, the effect of flow incidence on the total pressure losses is less pronounced. They swore that the large increase in losses at transonic conditions is due to a strong shock/boundary layer interaction that can lead to flow separation at the blade suction surface

The effect of the incidence angle on the boundary layer evolution of a low-pressure turbine blade was studied by [23] with the use of large vortex simulation. Five incidence angles on the blade were studied +7.8°, +5°, 0°, −5°, and −10°. Their calculation results showed that the influence of the incidence angle on the flow field is mainly visible at the front of the suction side and the pressure side. When the incidence angle changes from positive to negative, the separation bubble near the leading edge disappears and the blade load gradually decreases. When the incidence angle decreases to −5°, a separation bubble appears near the leading edge of the pressure side. In the case of an incidence angle equal to −10°, the length of the separation bubble increases to 39% of the axial chord.

Our work consists of giving a prediction by numerical simulation of a turbulent flow that crosses one of the blades of a turbine for different orientation angles. The simulation is carried out using a CFD calculation code. The numerical simulation is based on the finite volume method for the discretization of differential equations using the k-epsilon turbulence model. The equations that govern the flow are solved by the SIMPLE algorithm. The simulation allows us to observe the behavior of the fluid when crossing these blades for different angles of incidence (60°, 70° and 80°). The performance of the turbine depends mainly on the characteristics and efficiency of the blades for an optimal orientation angle position. The results obtained allow us to identify and characterize the flow around the blade by plotting the dynamic field, the pressure coefficient, and the Mach number.

2 Mathematical Modeling

The k-epsilon model is the most commonly used for the prediction of turbulent flows, it was proposed by [24]. This model is based on the addition of two additional equations one for the turbulent kinetic energy (k) and the other for the dissipation (epsilon), the k-epsilon model uses the Boussinesq approximation this hypothesis corresponds to link the Reynolds constraints to the mean gradients of turbulent velocity and viscosity.

$$-\rho\overline{u_i'u_j'} = \mu_t\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}\right) - \frac{2}{3}\rho k\delta_{ij} \quad (1)$$

where k is the turbulent kinetic energy, defined as:

$$k = \frac{1}{2}\overline{u_i'u_i'} \quad (2)$$

The turbulent viscosity is modeled as follows:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \quad (3)$$

with ε is the dissipation rate given by:

$$\varepsilon = \frac{\mu_t}{\rho}\left(\overline{\frac{\partial u_i}{\partial u_j}\frac{\partial u_i}{\partial u_j}}\right) \quad (4)$$

For the k-epsilon model the two additional equations are given:

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_i}(\rho k \bar{u}_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k + G_b - \rho\varepsilon - Y_M + S_k \quad (5)$$

$$\frac{\partial}{\partial t}(\rho\varepsilon) + \frac{\partial}{\partial x_i}(\rho\varepsilon\bar{u}_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{\varepsilon 1}\frac{\varepsilon}{k}(G_k + C_{\varepsilon 3}G_b) - C_{\varepsilon 2}\rho\frac{\varepsilon^2}{k} + S_\varepsilon \quad (6)$$

where

Gk and Gb are the turbulent kinetic energy due to mean velocity gradients and buoyancy, respectively.

$C_{\varepsilon 1}$, $C_{\varepsilon 2}$, and $C_{\varepsilon 3}$ are the constants given in Table 1.

σ_k and σ_ε are the turbulent Prandtl numbers for k and respectively given in Table 1.

S_k and S_ε are the source terms for k and ε respectively.

The turbulent kinetic energy due to mean velocity gradients is modeled as follows:

$$G_k = -\rho\overline{u_i'u_j'}\frac{\partial u_i}{\partial x_i} \quad (7)$$

The turbulence kinetic energy due to buoyancy is given by:

$$G_b = \beta g_i \frac{\mu_t}{\mathrm{Pr}_t}\frac{\partial T}{\partial x_i} \quad (8)$$

Where β is the coefficient of thermal expansion.

$$Y_M = 2\rho\varepsilon M^2 \quad (9)$$

Where M is the turbulent Mach number

The coefficients of the models are given in Table 1:

Table 1. Constants of the standard k–ε model

C_μ	$C_{\varepsilon 1}$	$C_{\varepsilon 2}$	$C_{\varepsilon 3}$	σ_k	σ_ε
0, 09	1, 44	1, 92	0.5	1, 0	1, 3

3 Geometry and Mesh

The study of the flow around the blade profiles of a turbine is presented in Figure 1 based on a case already studied by [25].The cascade profiles are called SE 1050 with a blade length of 1085 mm. The profile is a section of a rotor blade at a distance of 320 mm from the root; the different parameters are given in Table 2.

The geometry of the simulation domain and the blades was carried out with the Gambit software, whose meshes are generated in a very precise way in order to properly test our numerical model and to correctly verify the results obtained with the experimental model. The meshes are triangular and are well refined in the areas near the blade walls Figure 2. The mesh is divided into two parts, one area for the flow part and one area for the blade part. The mesh has a triangular shape of 28820 cells and 14771 nodes.

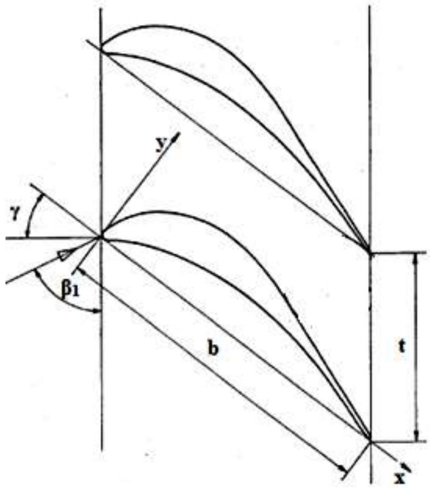

Fig. 1: Geometric of the turbine blades

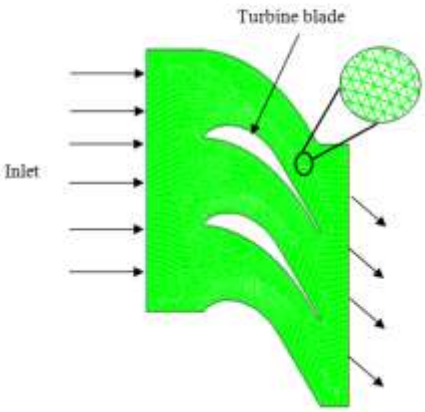

Fig. 2: Mesh generation

Table 2. Blade parameters

chord b	100 mm
blade span	160 mm
pitch t	55.12 mm
stagger angle) γ	37.11 deg
inlet angle ß1	70.7 deg

4 Boundary Conditions

The boundary conditions were chosen for four boundaries. At the flow inlet the Mach number is imposed M_1= 0.37, the pressure P_1= 98071.7 Pa and at the flow outlet M_2= 1.198.

5 Results and Discussions

Blade orientation is one of the most important elements of turbomachinery to increase the efficiency and operational reliability of turbines and compressors. Higher efficiency and operational reliability of turbines and compressors can be achieved by numerical modeling of the flow through the turbine blades

In this section, numerical results of a turbulent flow passing through two blades of a turbine for different orientation angles are presented using a computational fluid dynamics (CFD) code. The turbine's performance mainly depends on the characteristics and efficiency of the blades for an optimal orientation angle position. The numerical simulation is based on the finite volume method, and the discretization of the differential equations is solved by the SIMPLE algorithm. Different schemes have been used to solve these equations, such as the first upwind, the second upwind and the power law. The turbulence model used in this study is the standard k-epsilon model.

Figure 3 shows the pressure variation profile with respect to the inlet pressure at the blade walls obtained by the k- epsilon model compared with the experimental one. The results obtained by the numerical model are satisfactory with a small underestimation at the extrados and the trailing edge in the lower part. It is observed that the pressure is high in the upper part of the blade (extrados) and lower in the lower part (intrados).

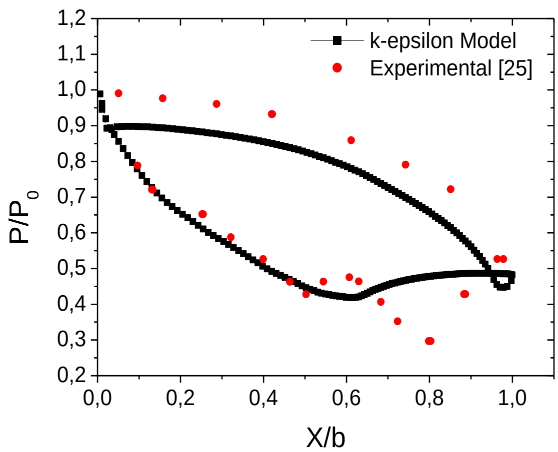

Fig. 3: Pressure profile

Figure 4, Figure 5 and Figure 6 show the contours of the pressure coefficient field obtained with the numerical model for the three blade orientation angles (60°, 70°, and 80°). It can be seen that the pressure coefficient has minimum values at the trailing edge and that its decrease remains a function of the blade orientation, the lower the blade orientation angle, the more the decrease in the pressure coefficient gradually approaches the leading edge on the upper part of the blade (extrados) and conversely, the greater the blade orientation, the more the decrease in the pressure coefficient is concentrated only at the trailing edge on the lower part of the blade (intrados). These observations can lead us to properly control the pressure depression just at the turbine blade outlet.

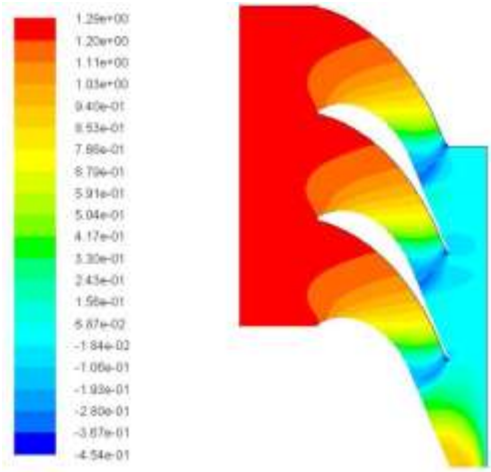

Fig. 4: Pressure coefficient contour (β=80°)

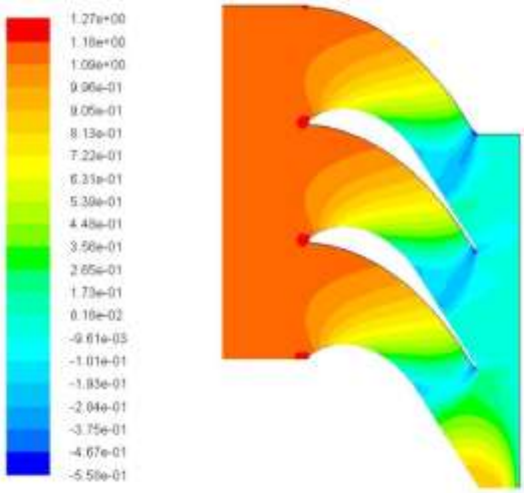

Fig. 5: Pressure coefficient contour (β=70°)

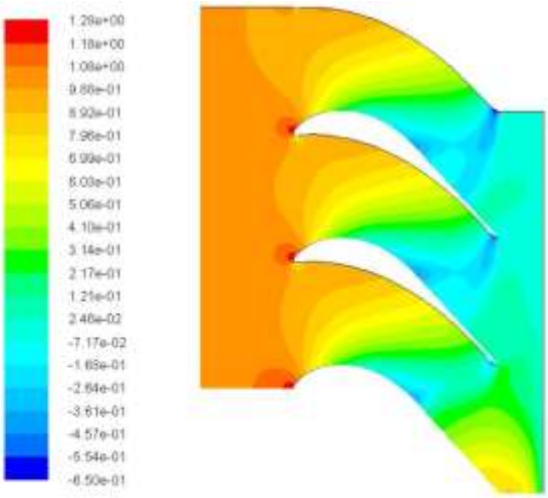

Fig. 6: Pressure coefficient contour (β=60°)

Figure 7, Figure 8 and Figure 9 show the contours of the Mach number obtained with the numerical model for the three blade orientation angles (60°, 70° and 80°). It can be seen that the Mach number has low values at the flow inlet, where the regime is subsonic. When the fluid crosses the passage sections between the blades, the flow regime becomes supersonic at the trailing edges in the lower part of the blade (intrados) and its value depends on the blade orientation. It can also be observed that the flow is supersonic in an almost limited area in the minimum section in the inner part between the blades and at the flow outlet where the blade orientation angle is 80°. For the case where the blade orientation angle is lower, that’s the case of 70° and 60°.The flow regime almost encompasses the upper part of the blade (extrados) and intensifies at the trailing edge where the flow regime approaches a maximum supersonic flow which causes more intense shock waves and which can cause deterioration of the turbine blades if it is not well controlled.

Figure 10, Figure 11 and Figure 12 show the contours of the dynamic field obtained with the numerical model for the three blade orientation angles (60°, 70° and 80°). In these figures, we observe a significant dynamic field that occurs at the trailing edges in the lower part of the blade (intrados) and its value also depends on the orientation of the turbine blades with almost the same observation as the case of the Mach number contour. We observe a dynamic field that is more intense when the blade orientation angle is smaller inversely to the pressure field; this observation clearly shows the application of Bernoulli's theory on fluid mechanics.

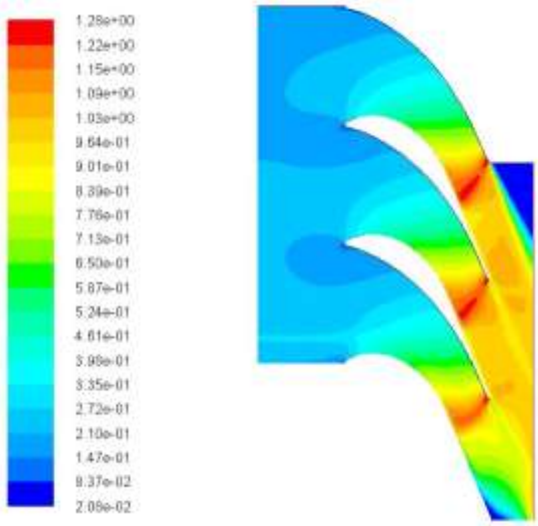

Fig. 7: Mach number contour (β=80°)

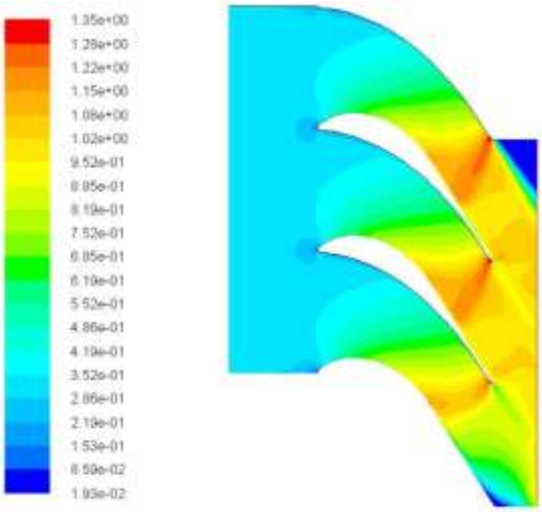

Fig. 8: Mach number contour (β=70°)

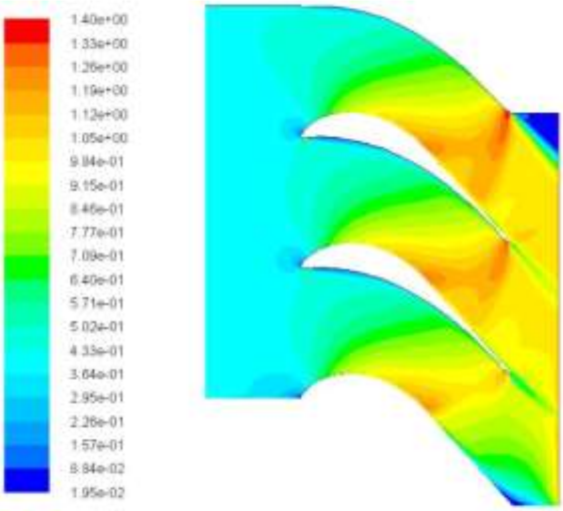

Fig. 9: Mach number contour (β=60°)

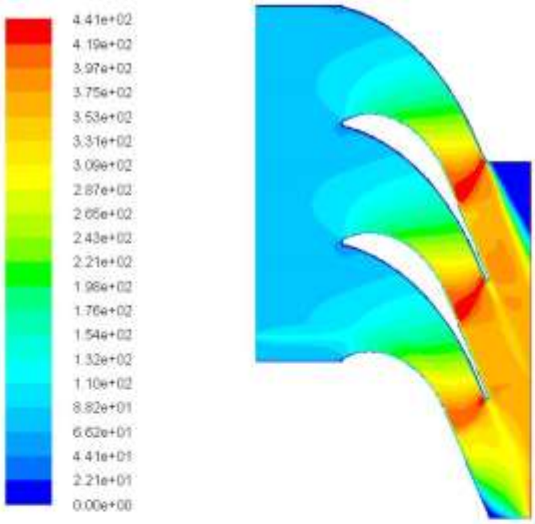

Fig. 10: Velocity contour (m/s) for (β=80°)

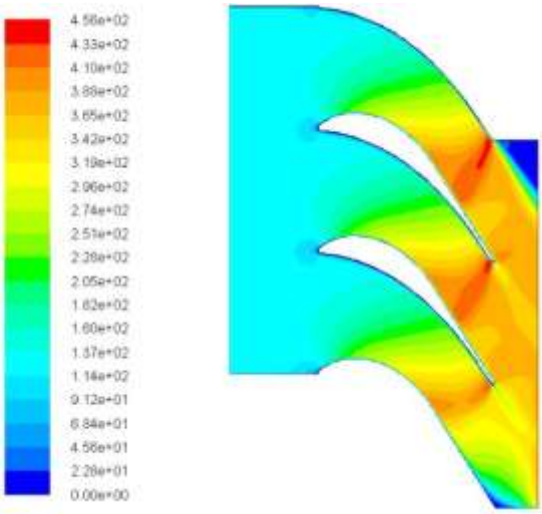

Fig. 11: Velocity contour (m/s) for (β=70°)

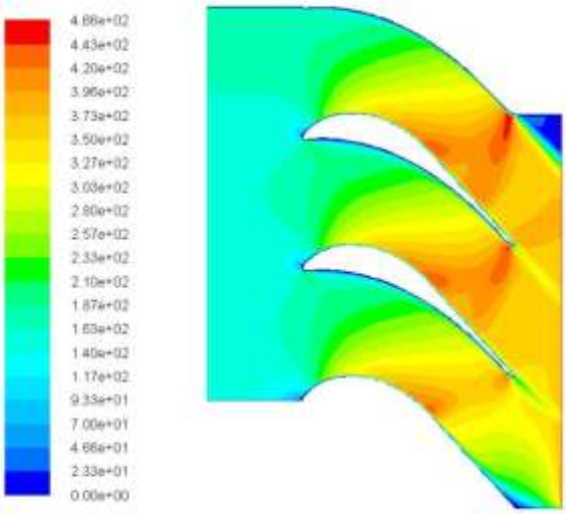

Fig. 12: Velocity contour (m/s) for (β=60°)

6 Conclusion

The objective of this work allowed us to study the effect of blade orientation on the behavior of a flow around the blades of a steam turbine using the numerical approach. The numerical approach was tested and the results obtained are very interesting and encouraging. The results obtained allowed us to identify and characterize the flow around the blades by plotting the dynamic field of the speed, the pressure coefficient and the Mach number. It was observed that the pressure coefficient has minimum values at the trailing edge and its decrease remains a function of the blade orientation, the lower the blade orientation angle, the more the decrease in the pressure coefficient gradually approaches the leading edge on the upper part of the blade (extrados) and conversely, the greater the blade orientation, the more the decrease in the pressure coefficient is concentrated only at the trailing edge in the lower part of the blade (intrados). When the fluid passes through the passage sections between the blades, the flow regime becomes supersonic at the trailing edge in the lower part of the blade (intrados) and its value depends on the orientation of the blades, the lower the orientation angle of the blades, the closer the flow regime is to a supersonic regime causing a shock wave. These observations can be very interesting points in the case of application in turbines, where we can control the pressure depression and the shock wave well just at the inlet and outlet of the turbine blades. This study can be followed by another future research that focuses on acoustic noise around turbine blades

References:

[1] Lei Chen, Bing Li, Zhuangde Jiang, Inspection of assembly error with effect on throat and incidence for turbine blades, *Journal of Manufacturing Systems*, Vol. 43, Part 3, 2017, pp. 366-374. https://doi.org/10.1016/j.jmsy.2017.03.007.

[2] Cho S-Y, Choi S-K. Experimental study of the incidence effect on rotating turbine blades. *Proceedings of the Institution of Mechanical Engineers, Part A, Journal of Power and Energy*, Vol. 218, No. 8, 2004, pp. 669-676. https://doi.org/10.1243/0957650042584384.

[3] Hoseinzade D, Lakzian E, Dykas S, Optimization of the trailing edge inclination of wet Steam turbine stator blade towards the losses reduction, *Experimental Techniques,* Vol. 47, 2023, pp.269-279. https://doi.org/10.1007/s40799-021-00534-5.

[4] J. Halama, J. Dobeš, J. Fořt, J. Fürst, K. Kozel, Numerical modeling of unsteady flow in steam turbine stage, *Journal of Computational and Applied Mathematics*, Vol. 234, No. 7, 2010, pp. 2336-2341. https://doi.org/10.1016/j.cam.2009.08.090.

[5] Luxa M, Příhoda J, Šimurda D . Investigation of the compressible flow through the tip-section turbine blade cascade with supersonic inlet, *Journal of Thermal Science*, Vol.25, 2016, pp.138–144. https://doi.org/10.1007/s11630-016-0844-0.

[6] Kafaei A, Salmani F, Lakzian E, Wróblewski W, Vlaskin M S, Deng Q, The best angle of hot steam injection holes in the 3D steam turbine blade cascade, *International Journal of Thermal Sciences*, Vol. *173*, 2022, 107387. https://doi.org/10.1016/j.ijthermalsci.2021.107387.

[7] J. Perez-Rodriguez, J. Sierra-Del Rio, L. F. Grisales-Noreña, S. Galvis, Optimization of the Efficiency of a Michell–Banki Turbine Through the Variation of Its Geometrical Parameters Using a PSO Algorith, *WSEAS Transactions on Applied and Theoretical Mechanics*, Vol. 16, 2021, pp. 37-46. https://doi.org/10.37394/232011.2021.16.5.

[8] Perdichizzi A, Dossena V, Incidence angle and pitch–chord effects on secondary flows downstream of a turbine cascade, *ASME. Journal of Turbomachinery*, Vol.115, No.3, 1993, pp.383-391. https://doi.org/10.1115/1.2929265.

[9] Mamaev B. I, Ermolaev G. V, Study of Turbine Cascades at Negative Incidence Angles, *Thermal Engineering*, Vol. 70, No. 3, 2023, pp. 215-222. https://doi.org/10.1134/S0040601523030035.

[10] Norhan M. Mousa, Yasser I. El-Shaer, Mohamed I. Abu El-Sebah, A Proposed Controller for Pitch Angle of Wind Turbine, *WSEAS Transactions on Systems and Control*, vol. 18, 2023, pp. 527-539. https://doi.org/10.37394/23203.2023.18.55.

[11] Gao Z, Narzary D, Mhetras S, Han J, Effect of Inlet Flow Angle on Gas Turbine Blade Tip Film Cooling. *ASME. Journal of Turbomachinery*, Vol. 131, No. 3, 2009, 031005. https://doi.org/10.1115/1.2987235.

[12] Merouane Habib, Wind Velocity Effect on the Aerodynamic and Acoustic Behavior of a Vertical Axis Wind Turbine, *WSEAS Transactions on Fluid Mechanics*, vol. 19, 2024 pp. 166-174. https://doi.org/10.37394/232013.2024.19.17.

[13] Efstathios E, Theotokoglou, Georgios Xenakis, GE 1.5 XLE Wind Turbine Blade Analysis with Computational Methods for Various Composite Materials, *WSEAS Transactions on Applied and Theoretical Mechanics*, 2023, vol 18, pp. 56-67. https://doi.org/10.37394/232011.2023.18.6.

[14] Li S.M, Chu T.L, Yoo Y.S, Ng W. F,Transonic and Low Supersonic Flow Losses of Two Steam Turbine Blades at Large Incidences. *Journal of Fluids Engineering*, Vol.126, No.6, 2004, 966. https://doi.org/10.1115/1.1839927.

[15] Feng Z, Sun R, Cui W, Xu J,Research on Effects of Incidence to Turbine Guide Cascade Aerodynamic Performance. *Tehnički vjesnik*, Vol. 26, No. 3, 2019, pp. 736-742. https://doi.org/10.17559/TV-20190129032154.

[16] Ruifeng Zhang, Weihao Zhang, Mingyu Gan, Wei Xiao, Fei Zeng, Duanbing Zhao, Toward high-lift low-solidity design for incidence tolerant gas turbine blade profile, *Energy*, Vol. 309, 2024, 133034. https://doi.org/10.1016/j.energy.2024.133034.

[17] Chen Y, Cai L, Jiang D, Li Y, Du Z, Wang, S, Experimental and numerical investigations for turbine aerodynamic performance with different pressure side squealers and incidence angles. *Aerospace Science and Technology*, Vol. 136, 2023, 108234. https://doi.org/10.1016/j.ast.2023.108234.

[18] McVetta A, Giel P, Welch G, Aerodynamic investigation of incidence angle effects in a large scale transonic turbine cascade. *In 48th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit Atlanta, Georgia*, 2012, p. 3879. https://doi.org/10.2514/6.2012-3879.

[19] Liu C, Zhu H, Fu Z, Xu R, The Effects of Inlet Reynolds Number, Exit Mach Number and Incidence Angle on Leading Edge Film Cooling Effectiveness of a Turbine Blade in a Linear Transonic Cascade. *Proceedings of the ASME Turbo Expo 2015: Turbine Technical Conference and Exposition*, *Volume 5B: Heat Transfer*. Montreal, Quebec, Canada. June 15–19, 2015. https://doi.org/10.1115/GT2015-42888.

[20] Jie Gao, Qun Zheng, Ping Dong, Weiliang Fu,Effects of flow incidence on aerothermal performance of transonic blade tip clearance flows, *Energy*,Vol.139, 2017, pp. 196-209. https://doi.org/10.1016/j.energy.2017.07.175.

[21] Chibli, HA, Abdelfattah, SA, Schobeiri, MT, Kang, C, An Experimental and Numerical Study of the Effects of Flow Incidence Angles on the Performance of a Stator Blade Cascade of a High Pressure Steam Turbine." *Proceedings of the ASME Turbo Expo 2009: Power for Land, Sea, and Air*. *Volume 7, Turbomachinery, Parts A and B*. Orlando, Florida, USA. June 8–12, 2009. pp. 821-830. https://doi.org/10.1115/GT2009-59131.

[22] Yoo S. J, Ng, W. F, Influences of mach number and flow incidence on aerodynamic losses of steam turbine blade. *KSME international journal*, Vol. *14*, 2000, pp. 456-465. https://doi.org/10.1007/BF03186440.

[23] Wang Y, Sui X, Zhang K, Xiang X, Zhao Q, Effects of incidence angle on a low-pressure turbine blade boundary layer evolution through large eddy simulation. Proceedings of the Institution of Mechanical Engineers, Part A, *Journal of Power and Energy*, Vol.232, No.6, 2018, 722-734. https://doi.org/10.1177/0957650917752806.

[24] B. E.Launder, G. J.Reece, W.Rodi, Progress in the Development of a Reynolds-Stress Turbulent Closure, *Journal of Fluid Mechanics*, Vol. 68, 1975, pp. 537-566. https://doi.org/10.1017/S0022112075001814.

[25] Šafařík P, Experimental data from optical measurement tests on a transonic turbine blade cascade, *Proc. of the 13th Symposium on Measuring Techniques for Transonic and Supersonic Flow in Cascades and Turbomachines* (eds. C. Grossweiler, G. Gyarmathy), 20/0-20/14, ETH Zürich, 1996.

Contribution of Individual Authors to the Creation of a Scientific Article (Ghostwriting Policy)

The authors contributed to the present research, at all stages from the formulation of the problem to the final findings and solution.

Sources of Funding for Research Presented in a Scientific Article or Scientific Article Itself

No funding was received for conducting this study.

Conflict of Interest

The author has no conflicts of interest to declare.