International Journal on
"Technical and Physical Problems of Engineering"
(IJTPE)
Published by International Organization of IOTPE

ISSN 2077-3528
IJTPE Journal
www.iotpe.com
ijtpe@iotpe.com

September 2024 | Issue 60 | Volume 16 | Number 3 | Pages 135-141

FLOW DYNAMICS AND COOLING OPTIMIZATION IN A DISCOIDAL MACHINE ALTERNATOR WITH MULTIPLE JETS: INTEGRATED APPROACH AND NUMERICAL SIMULATIONS

A. El Hannaoui [1,2] R. Boutarfa [1,2] C. Haidar [1,2]

1. Laboratory of Engineering, Industrial Management and Innovation (IM2I), Department of Mechanical Engineering, Faculty of Science and Technology, Hassan I University, Settat, Morocco
2. Department of Energy Laboratory, National School of Arts and Crafts (ENSAM), University of Moulay Ismail, Meknes, Morocco, a.elhannaoui@uhp.ac.ma, boutarfarachid@yahoo.fr, chadia.haidar@gmail.com

Abstract- This paper explores the influence of multiple jets on the fluid dynamics in the magnetic air zone of a disk alternator., focusing on cooling effects. Using numerical simulations with ANSYS Fluent software. More precisely, the analysis range extends from the rotational Reynolds number up to 5.14×10^5, while varying the jet Reynolds value in a band from 16×10^3 up to 49.5×10^3. The variation in G spacing, between 0.02 and 0.04, is a key element in our study, significantly influencing the flow dynamics and thermal characteristics within the system studied. The study aims to optimize the cooling of a disk by merging center-jet and eccentric-jet cooling schemes. The results show a significant improvement in average heat transfer with the addition of jets, demonstrating the effectiveness of the integrated approach. The analysis of velocity profiles and Nusselt numbers highlights phenomena such as the formation of vortices promoting heat transfer, and the influence of rotation on flow dynamics. This study offers promising prospects for the design of mechanical systems, helping to optimize the energy performance and operational stability of the equipment studied.

Keywords: Heat Transfer, Rotor-Stator, Impacting Jets, RSM Turbulence Model.

1. INTRODUCTION

Flow between the moving and stationary elements of mechanical systems such as wind turbines, compressors and aircraft engines are frequent and crucial. In the field of fluid mechanics, this flow manifests itself in the confined spaces shown in (Figure 1). However, when fluid mechanics is applied exhaustively, the level of precision of calculations decreases inside these cavities. By simulating this complex flow between rotor and stator, it becomes possible to analyze the flow's internal characteristics precisely, and to capture subtle variations more easily. Moreover, this type of flow in the cavity between the elements has a considerable influence on the system's overall efficiency and dependability.

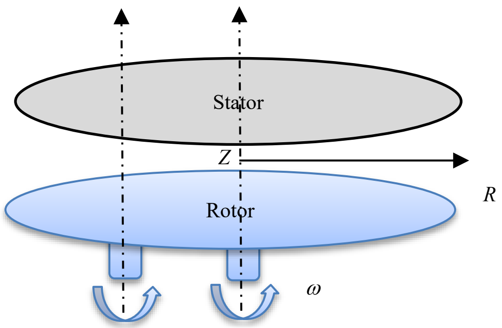


Figure 1. Rotating and stationary elements

Various investigations have examined the flow in the internal space of the device, highlighting specific approaches to quantifying the critical Reynolds number, notably those of [1]. Provided detailed descriptions of flow around a rotating disk, while, identified four flow regimes as a function of disk speed and rotor-stator spacing [2, 3]. Recently carried out an in-depth experimental study of the flow characteristics. Their results revealed a centripetal flow near the contact zone [4]. The measurements indicate a limited influence of rotation on the fluid flow at contact points, but for larger radii, rotation predominates, transforming the flow into a centrifugal one.

Have extended the experimental work started by Ng [4], comparing their data with the predictions of turbulence models (k - ω SST and RSM). In parallel, J. Pelle, et al. [6] examined heat exchange in a stator rotor assembly incorporating an axisymmetric air jet, putting forward proposed relationships for average Nusselt numbers of a various measured values of G [5, 6]:

- For $0.01 < G < 0.02$; $\overline{Nu} = 0.08G^{-0.07}\,\mathrm{Re}_j^{\;0.5}\times R_e^{\;0.25}$

- For $0.04 < G < 0.08$; $\overline{Nu} = 0.006G^{0.15}\,\mathrm{Re}_j^{\;0.5}\times R_e^{\;0.5}$

- For $G = 0.16$; $\overline{Nu} = \mathrm{Re}_j^{\;0.25}\times R_e^{\;0.5}$

Irrespective of the value of the spacing G, research by *J.* Revealed the existence of a unique ridge on the radial profile of the (Nur) [6].

Registered: 240111 Article Info: 1841 Accepted: 240823
Reserved: 240414 Available: 240930

As part of his study, [7] undertook direct numerical simulations (DNS) to examine the ventilation process of a rotational plate exposed to a jet. The geometric parameters used are in line with the experiments carried out by [6]. The conclusions of this research are based on the analysis of radial distributions of mean speed components and associated Reynolds stress tensors. The distribution of the Nusselt number, an indicator of heat exchange, reveals two distinct peaks. The first is attributed to high fluid velocities, generating significant shear stresses. The second peak, consistently positioned with a uniform amplitude for all rotation configurations. It should be noted that an increase in the rotation parameter generally leads to a concomitant rise in the mean Nusselt number. These observations suggest that plate rotation exerts an influence on heat transfer, with an increase in rotation being associated with an improvement in cooling efficiency.

Undertook an integrated approach combining numerical simulations and practical experiments to explore the cooling process of a spinning rotor exposed to an eccentrically impacting jet, in a configuration similar to that developed by all [6, 8]. The results obtained are presented in the form of velocity and Nur. Our study aims to merge the central jet cooling approach of R. Oguic and the eccentric jet approach of Chadia, et al. to design a novel integrated method. Based on numerical simulations, this method aims to optimize the cooling of a disk by combining two distinct cooling schemes, as illustrated in Figure 2. Our goal is to achieve a more efficient and complete cooling system for the disk by leveraging the advantages of each approach while overcoming their respective limitations.

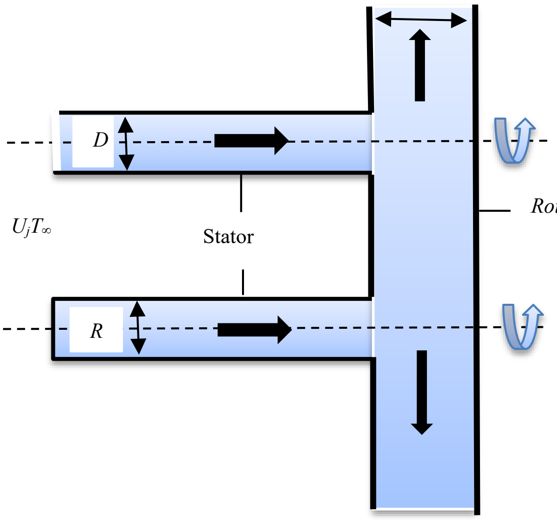


Figure 2. Improved cooling of rotor-stator geometry by multiple impact jets

2. NUMERICAL SIMULATION

Our modeling approach is based on the use of numerical simulation via ANSYS Fluent software. We plan to exploit the advanced features of this recognized tool in the field of computational fluid mechanics. ANSYS Fluent offers a robust and versatile platform for efficiently solving equations of motion and heat transfer. This ANSYS Fluent simulation will enable us to study in detail the complex dynamics of flow and heat exchange, offering valuable perspectives for our research.

2.1. Definition of the Analysis Perimeter: Boundary Conditions

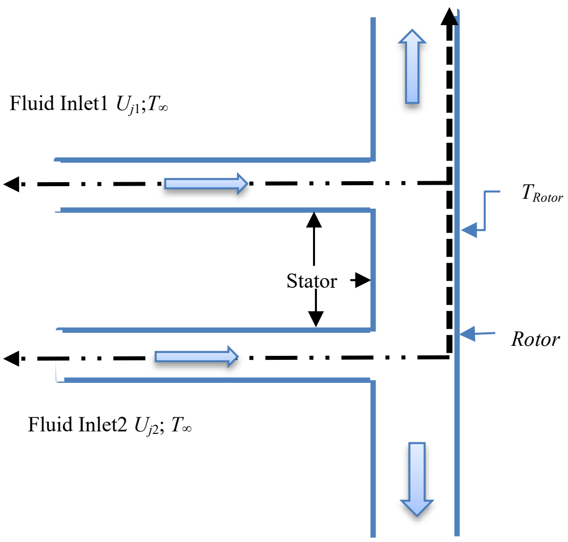
The boundary conditions for this simulation, defined for a disk with diameter R_d equal to 320 mm, encompass several key aspects. Air jet characteristics, including inlet parameters, are determined for each jet. This includes the velocity (Re_j) and temperature (T_j) configured for the air inlet on the disc surface. The inlet velocity is adjusted to match the desired velocity profile, while the inlet temperature is adapted to reflect the thermal properties of the incoming air.

In addition, outlet conditions are specified to define the static pressure or outlet velocity of the air after its interaction with the disc. In parallel, a thermal condition is established on the straight surface of the disc, with a temperature (Tr) fixed at 80 degrees, representing the specific thermal characteristics of this zone. The variable rotation speeds of the disk are associated with the definition of the rotational Reynolds number, an essential quantity for modeling flow as a function of different rotation speeds. All these boundary conditions are crucial to the accurate simulation of disk cooling by the two air jets in ANSYS Fluent.

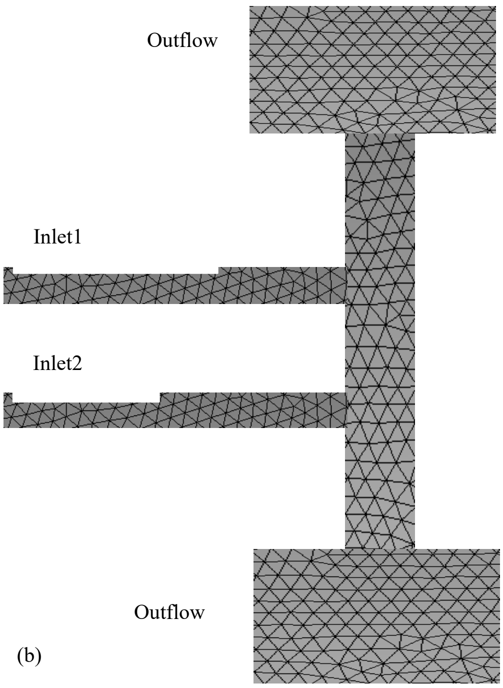
- The Reynolds number of the flowing stream $\mathrm{Re}_j = \frac{U_j \times D}{\nu}$.
- The rotational Reynolds number $\mathrm{Re}_\omega = \frac{\omega \times R_d^2}{\nu}$.

The boundary conditions (Figures 3a and 3b) defined for this numerical simulation are based on the use of a tetrahedral mesh. This type of mesh, made up of tetrahedrons, provides a flexible representation of complex geometries, well suited to fluid mechanics simulations. In addition, the RSM (Reynolds Stress Model) turbulence model was selected for this study. RSM, which is more advanced than simpler turbulence models such as *k*-*ε*, enables Reynolds stresses to be modeled, which is crucial for more complex flows. As far as inlet conditions are concerned, the Reynolds number, noted Re_j, is decisive. These inlet conditions for velocity and turbulence describe how air is introduced into the simulation domain. By combining a tetrahedral mesh, the RSM model and the inlet conditions in terms of Re_j, this model seeks to obtain an accurate and detailed representation of flow phenomena studied, particularly for complex configurations.

In this context, the characteristics of the jets used are determined by the rotational number, varying between 16×10^3 and 49.5×10^3, inducing different dynamics in the flow. In addition, the range of Reynolds number variation extends from 2.58×10^5 to 5.14×10^5, characterizing turbulent flow conditions through the cavity. At the same time, the spacing between the rotor and stator, parameterized by the value of *G*, oscillates between 0.02 and 0.1, significantly affecting the interactions and flow patterns between these two elements. These multiple and varied values represent a diversity of operating conditions, enabling us to explore a wide range of flow configurations in the cavity of the rotor-stator system.



(a)



(b)

Figure 3. a) Flow layout, b) Mesh diagram

3. MODELING APPROACHES FOR TURBULENCE IN NUMERICAL SIMULATIONS

Our approach is based on the use of a single-point statistical model incorporating a second-order closure with a low Reynolds number, derived from the model of [9]. This approach also takes into account the effects of rotation [11]. The model we apply, RSM (Reynolds Stress Model), offers a detailed representation of the turbulent field in the entire domain, including near the walls, and has the advantage of not requiring an assumption of turbulence viscosity. The general equation for the components of the Reynolds tensor, denoted R_{ij}, is formulated in a relative reference frame:

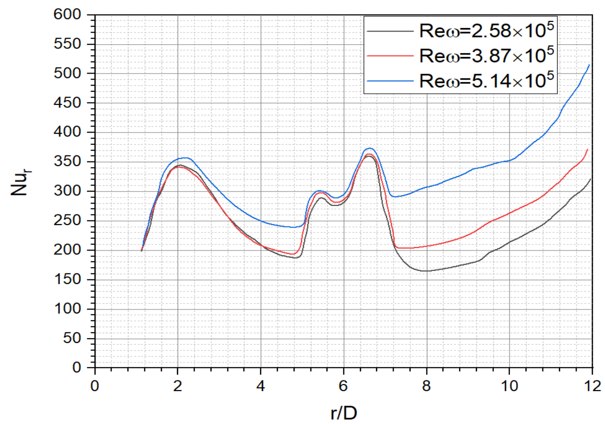
$$\bar{u}_j \frac{\partial \overline{u_i' u_j'}}{\partial x_j} = P_{ij} + D_{ij} - \varepsilon_{ij} + O_{ij} + \Omega_{ij} + \Phi_{ij} \tag{1}$$

In the context of the general equation for Reynolds tensor components (R_{ij}) formulated in a relative frame of reference, the corresponding terms P_{ij} ; Ω_{ij} ; φ_{ij} ; ε_{ij} and O_{ij} are represented simultaneously as the diffusion term, the pressure deformation correction term and the dispersion term. These components work together to describe the behavior of the turbulent field, taking into account diffusion, pressure deformation correction and dispersion.

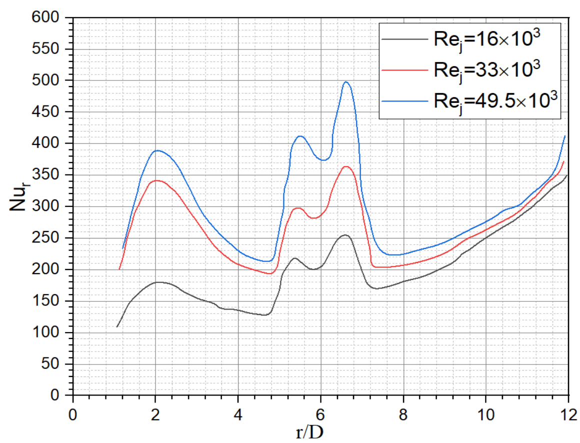
4. RESULTS AND DISCUSSION

In this configuration, characterized by small spacing and a fixed jet-relative Reynolds number Re_j (Figure 4a), local Nur increase with increasing disk radius. Their dependence on the rotational Reynolds number is limited, close to center. Near the collision points, the effect of airflow is more pronounced. The radial evolution of the Nur reveals three peaks not far from the contact surface. The first peak, associated with the central jet, is located at distance r/D= 2, while other two peaks are at 5.2 and 6.8.

Moving away from the point of impact, the Nur curves associated with the different rotation speeds show a continuous increase. Comparing these results with the observations of [5], [6], and [10] for a central jet, and the work of [8] for an eccentric jet, we see that cooling takes place at the center of the disk.



(a)



(b)

Figure 4. a) Nusselt number for Re_j=33000 and for G=0.04, b) Nusselt number for Re_w=387000 and for G=0.04

For Re_ω (Figure 4b) and different values of Re_j, local Nur show an accelerating trend with increasing jet airflow. Injected air flows influence local convective heat dissipation. However, in the vicinity of the impact points, exchanges are particularly high, inducing a sharp increase in heat transfer. Thus, renewing the air and increasing heat exchange. These phenomena are reflected in peaks in heat transfer, the location and intensity of which vary as a function of Re_j.

Figure 5 of our study offers a visual representation of the evolution of the Nur as a function of the (Re_w), highlighting the significant impact of jet addition on these values. The Nusselt number, a dimensionless parameter, is used to characterize the efficiency of convective heat transfer. In our investigation, the incorporation of jets results in a substantial improvement in average heat transfer, as reflected by the average Nusselt number.

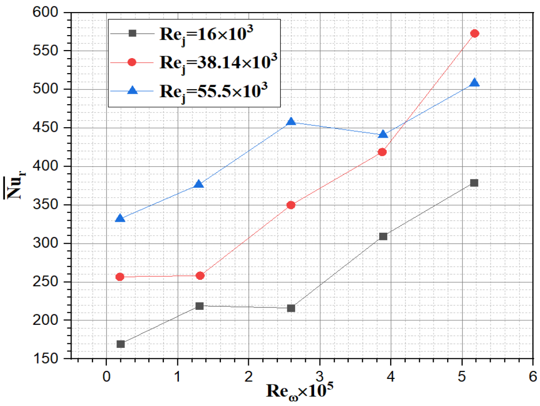


Figure 3. Analysis of Convective Heat Transfer Enhancement with Added Jets: Exploring the Average Nusselt Number as a Function of Rotational Reynolds

Next, our attention turns to exploring the sensitivity of the results to a variation in the rotational Reynolds number, Re_ω. This parameter, frequently used to characterize fluid rotation, will be studied in detail. The aim is to understand how fluid rotation influences heat transfer in the specific context of our research. This in-depth analysis of sensitivity to variations in the rotational Reynolds number will help shed further light on the mechanisms underlying heat transfer efficiency in the system under study.

In this study (Figure 6), we have undertaken in-depth characterization of velocity contour profiles as a function of various Re_j values (33×10^3 and 49.5×10^3), while holding Re_ω constant at 3.87×10^5. This investigation aims to understand the complex interactions between airflow, device rotation, and their impacts on temperature distribution. A major observation lies in the significant influence of rotation and airflow on the surrounding temperature. The results indicate that the highest temperature values occur at small radii and from the periphery of the rotor. This suggests a close correlation between flow configuration, device rotation and heat dissipation. Detailed analysis of the velocity contour profiles will provide a better understanding of heat transfer phenomena in this specific context, paving the way for more efficient applications and potential improvements in the design of the systems studied.

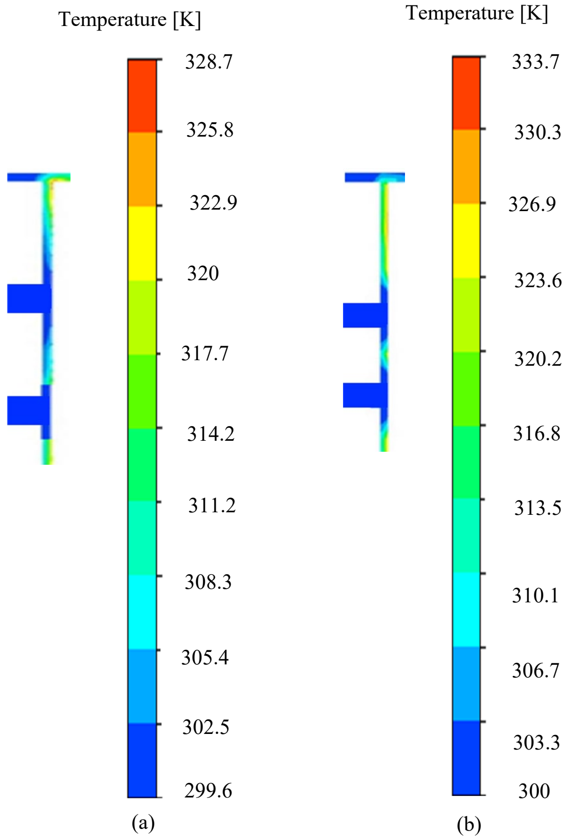


Figure 4. Characterization of Velocity Contour Profiles for Various Re_j Values a) 33×10^3, b) 49.5×10^3 at $Re_\omega=3.87\times10^5$

For $Re_\omega = 2.58\times10^5$ and $G = 0.04$ (Figure 7a1), the radial elements of the velocity, noted at various radial locations (r/D = 4.8, 5, 6.73, 6.92 and 7.11), show a parabolic profile similar to that obtained in the case of two parallel plates. The maximum peak of this magnitude is reached at the midpoint of the two disks for Z/H=0.5.

However, a negative radial velocity is observed for r/D=6.92 and 7.11, corresponding to fluid re-entry. At r/D=6.73, in the middle of the rotor between the two jets, a symmetrical radial velocity is observed. Tangential velocity values (Figure 7: a2), which describe the flow in the vicinity of the rotor, are notably high. However, they gradually decrease with distance from the rotating disk in the direction normal to the rotor.

Figure 7 of our study highlights the axis-to-axis distribution of radial and tangential velocities, considering non-zero rotation parameters, at three distinct positions for r/D (4.8, 5; 6.73; 6.92; 7.11). Examination of the characteristics of the radial and tangential velocity profiles enables us to identify different outflow structures. A stream with two clearly defined limiting levels, divided by a vigorously rotating, inviscid liquid, is categorized as a Batchelor Stream. Conversely, if the stream has only one boundary layer, with virtually zero tangential velocity at the distance of this boundary layer, it is referred to as a Stewartson stream. These distinctions underline the variability of flow phenomena as a function of rotation parameters and positions in the configuration under consideration.

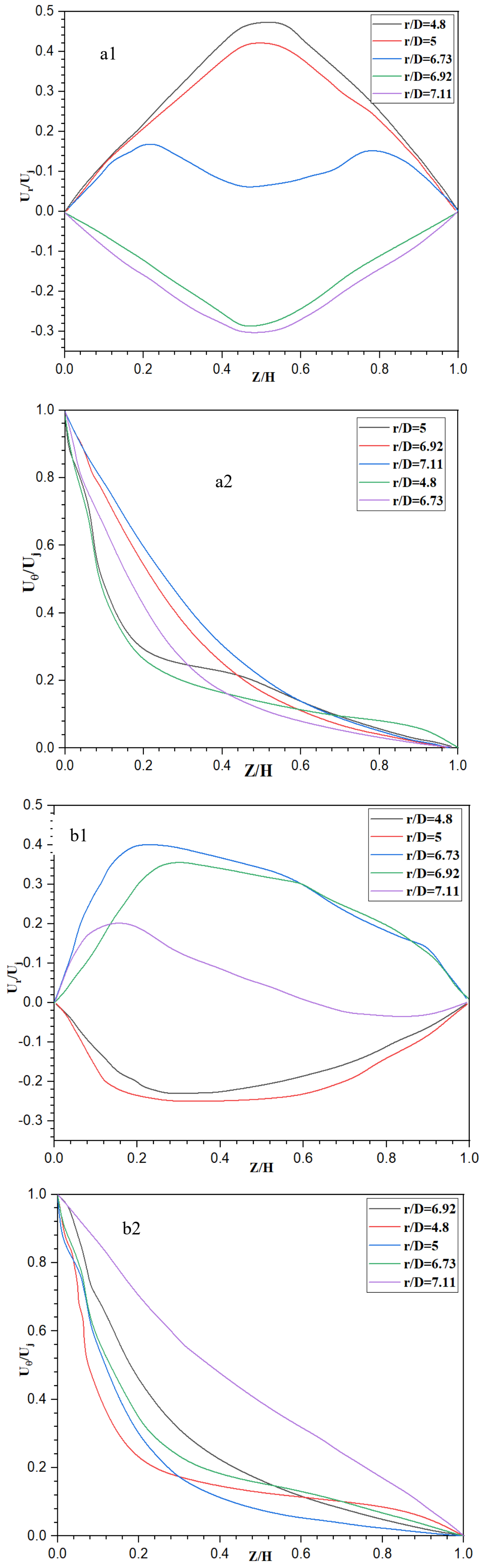


Figure 7. Study of the axial distribution of radial and tangential velocities in a flow around a cylinder for various radial positions

For $\mathrm{Re}_\omega = 2.58\times10^5$ and $G = 0.04$ (Figure 7a1), the radial elements of the velocity, noted at various radial locations (r/D = 4.8, 5, 6.73, 6.92 and 7.11), show a parabolic profile similar to that obtained in the case of two parallel plates. The maximum peak of this magnitude is reached at the midpoint of the two disks for Z/H=0.5. However, a negative radial velocity is observed for r/D=6.92 and 7.11, corresponding to fluid re-entry. At r/D=6.73, in the middle of the rotor between the two jets, a symmetrical radial velocity is observed. Tangential velocity values (Figure 7a2), which describe the flow in the vicinity of the rotor, are notably high. However, they gradually decrease with distance from the rotating disk in the direction normal to the rotor.

At the stator surface, characterized by Z/H=1, these values ultimately reach zero. This development highlights a significant variation in flux within the system, with initially high tangential values near the rotor gradually declining to negligible values at the stator surface. At the stator surface, characterized by Z/H=1, these values ultimately reach zero. This development highlights a significant variation in flux within the system, with initially high tangential values near the rotor gradually declining to negligible values at the stator surface.

Velocities in the circumferential direction are characterized by a single frontier zone close to the rotor. As one moves increasingly away from this boundary zone, the circumferential velocity decreases, finally reaching zero when z/h=1. The flow phenomena identified in this configuration are therefore similar to Stewartson-type [12] flows.

For $\mathrm{Re}_j = 49.5\times10^3$ and $\mathrm{Re}_\omega = 3.87\times10^5$ (see Figure 7 b), the axial evolution (along 'Z') of the radial component of the velocity for radii greater than $r/D > 6.73$ shows that the flow structure is closer to that of Batchelor type [13]. Centrifugal flow is observed near the rotor, and centripetal flow near the stator, with the presence of a boundary layer on the surface of each disk.

For radii smaller than $r/D < 6.73$, however, the flow remains centripetal throughout the air gap. The tangential component (Figure 7b2), of the velocity decreases as a function of z/H. Close to the rotor, centrifugal flow is mainly generated by the inertial force exerted on the fluid in the boundary layer linked to the rotating disk. This centrifugal radial flow is accelerated from the axis of rotation towards the periphery. Close to the stator, a centripetal flow (for r/D=4.8 and 5) forms in the stator boundary layer, where the flow takes on a spiral shape. This flow creates an inflow of fluid from the stator boundary layer to the rotor boundary layer. As a result, radial velocity increases in the rotor boundary layer and decreases in the stator boundary layer.

5. CONCLUSION

In conclusion, this study focuses on the influence of multiple jets on the cavity flow dynamics of a rotor-stator system, with particular emphasis on cooling effects. Previous research has explored various aspects of cavity flow, ranging from quantitative Reynolds number analyses to detailed experiments on flow characteristics. This paper proposes a novel approach by merging central jet and eccentric jet cooling schemes to design an integrated method. Using numerical simulations performed with ANSYS Fluent, the study aims to optimize the cooling of a disc by combining these two distinct cooling schemes.

The results reveal a substantial improvement in average heat transfer with the addition of jets, demonstrating the effectiveness of the integrated approach. Analysis of velocity profiles, Nusselt numbers and flow characteristics highlights phenomena such as the existence of vortices favoring heat transfer and the significant influence of rotation on flow dynamics. In summary, this study offers promising prospects for the design of mechanical systems, particularly in the fields of wind turbine engines, compressors and aircraft engines. By combining the advantages of central and eccentric jets, the integrated approach presented could lead to more efficient cooling systems, thus contributing to the optimization of energy efficiency and the operational stability of the equipment studied.

REFERENCES

[1] A.J. Faller, "An Experimental Study of the Instability of the Laminar Ekman Boundary Layer", Journal of Fluid Mechanics, Vol. 15, Issue 04, pp. 560-576, 1963.
[2] T.V. Van Karman, "On Laminar and Turbulent Friction", ZAMM-Journal of Applied Mathematics and Mechanics", Journal of Applied Mathematics and Mechanics, Vol. 4, pp. 233-252, 1921.
[3] J. Owen, C. Haynes, F. Bayley, "Heat Transfer from an Air-Cooled Rotating Disk", Proceedings of the Royal Society of London, A Mathematical and Physical Sciences, Vol. 336, Issue 1607, pp. 453-473, 1974.
[4] T.D. Nguyen, J. Pelle, S. Harmand, S. Poncet, "PIV Measurements of an Air Jet Impinging on an Open Rotor-Stator System", Exp. Fluids, Vol. 53, pp. 401-412, 2012.
[5] S. Poncet, T.D. Nguyen, S. Harmand, J. Pelle, R. Da Soghe, C. Bianchini, S. Viazzo, "Turbulent Impinging Jet Flow into an Unshrouded Rotor-Stator System: Hydrodynamics and Heat Transfer", Int. J. Heat Mass Transfer, Vol. 44, pp. 719-734, 2013.
[6] J. Pelle, S. Harmand, "Heat Transfer Study in a Rotor-Stator System Air-Gap with an Axial Inflow", Appl. Therm. Eng., Vol. 29, pp. 1532-1543, 2009.
[7] R. Oguic, S. Poncet, S. Viazzo, "High-Order Direct Numerical Simulations of a Turbulent Round Impinging Jet onto a Rotating Heated Disk in a Highly Confined Cavity", International Journal of Heat and Fluid Flow, Vol. 61, pp. 366-378, 2016.
[8] C. Haidar, R. Boutarfa, S. Harmand, "Fluid Flow and Convective Heat Transfer Analysis on a Rotor of Wind Turbine Alternator with an Impinging Jet", International Journal of Renewable Energy Research, Vol. 9, No. 3, 2019.
[9] B. Launder, D. Tselepidakis, "Application of a new Second-Moment Closure to Turbulent Channel Flow Rotating in Orthogonal Mode", International Journal of Heat and Fluid Flow, Vol. 15, pp. 2-10, 1994.
[11] A. Elhannaoui, C. Haidar, R. Boutarfa, "Convection Heat Transfer in an Air Gap in the Rotor/Stator System Subjected to a Central Jet", International Journal on Technical and Physical Problems of Engineering (IJTPE), Issue 53, Vol. 14, No. 4, pp. 260-265, December 2022.
[12] K. Stewartson, "On the Flow between two Rotating Coaxial Disks", Mathematical Proceedings of the Cambridge Philosophical Society, Cambridge University Press, Vol. 49, pp. 333-341, 1953
[13] G.K. Batchelor, "Note on a Class of Solutions of the Navier-Stokes Equations Representing Steady Rotationally-Symmetric Flow", Q.J. Mech. Appl. Math., Vol. 4, No. 1, pp. 29-41, 1951.

BIOGRAPHIES

Name: **Abdellatif**
Surname: **El Hannaoui**
Birthday: 09.01.1989
Birthplace: El Brouj, Morocco
Bachelor: Technical ND Industrial Science Preparatory Class Engineering, Mohammedia, Morocco, 2011
Master: Industrial Engineering, Department of Electrical Engineering Mechanical Engineering, Hassan I University, FST, Settat, Morocco, 2014
Doctorate: Student, Mechanical and Energy Engineering, Department of Industrial Engineering and Management, Faculty of Technical Sciences, Hassan I University, Settat, Morocco, Since 2021
Research Interests: Energy Mechanics
Scientific Publications: 1 Paper

Name: **Rachid**
Surname: **Boutarfa**
Birthday: 18.01.1973
Birthplace: Benselimane, Morocco
Bachelor: Energetic Mechanics, University of Valenciennes and Hainaut-Cambresis, France, 1994
Master: Mechanical and Energy Engineering, University of Valenciennes and Hainaut-Cambresis, France, 1997
Doctorate: Energy Engineering, University of Valenciennes and Hainaut-Cambresis, France, 2001
The Last Scientific Position: Prof., Mechanical Engineering Department, Faculty of Science and Technology, Hassan I University, Settat, Morocco, Since 2004
Research Interests: Numerical and Experimental Study of Convective Heat Transfer on Rotating Machines in Order to Improve the Energy Efficiency of Thermal Systems
Scientific Publications: 7 Papers

Name: **Chadia**
Surname: **Haidar**
Birthday: 03.03.1991
Birth Place: Fkih Ben Salah, Morocco
Bachelor: Science Physics, Preparatory Class, ENSA, Khouribgha, Morocco, 2011
Master: Energy and Environmental Process Engineering, Department of Process Engineering, Hassan I University, ENSA, Khouribga, Morocco, 2014
Doctorate: Energetic Mechanics, Hassan I University, FST, Settat, Morocco, 2020
The Last Scientific Position: Lecturer, Energy Department, Moulay Ismail University, ENSAM, Mekens, Morocco
Research Interests: Energetics, Heat Transfer
Scientific Publications: 6 Papers