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

December 2022 | Issue 53 | Volume 14 | Number 4 | Pages 260-265

CONVECTIVE HEAT TRANSFER IN AN AIR GAP OF A ROTOR-STATOR SYSTEM SUBJECTED TO A CENTRAL JET

A. El Hannaoui C. Haidar R. Boutarfa

Laboratory of Engineering, Industrial Management and Innovation, Department of Mechanics,
Faculty of Science and Technology, University of Hassan I, Settat, Morocco
a.elhannaoui@uhp.ac.ma, boutarfarachid@yahoo.fr, chadia.haidar@gmail.com

Abstract- This work represents a numeric simulation of heat transfer via convection in an airspace in a rotor/stator discoid technology. This rotor is cooled by a central impacting jet. The void is defined by a dimensionless distance *G*. The rotational Reynolds number varies from $Re_\omega = 2.29\times10^5$ to $Re_\omega = 4.69\times10^5$ and the jet-related Reynolds number ranges from 16.6×10^3 to 38.133×10^3. The numerical study is performed with the Ansys-Fluent computational code. The turbulence model chosen is RSM (Reynolds Stress Model). The influence of the rotation parameter on the heat transfers has been studied. Two distinct zones have been identified: one influenced by the air jet near the impact zone and characterized by high heat transfers, the other one the results of the numerical simulation are presented as mean axial and radial velocity and also as local and mean Nusselt number. The acquired results show a good agreement with the experiment result.

Keywords: Rotor-Stator, Impact Jet, Heat Transfer, Model RSM.

1. INTRODUCTION

Nowadays, the production of so-called "clean" [1] energy faces new challenges related to the needs of sustainable development, which is why the wind turbine has been developed recently because it can convert mechanical energy into electrical energy without producing greenhouse gases. Research on the optimization of these wind turbines and on-board generators has brought forward the "discoid" technology, which places a rotating disc in front of a fixed disc. The main drawback is that the flow of air produced by the rotation of this generator is not always sufficient for an optimal cooling of the installation. This is why This study focuses on the modification and exchange of air flows in the air space between these two disks. However, the main drawback of this generator is that the air flow that produces the rotation of this generator is not sometimes sufficient to optimize the cleaning of the installation. This study therefore focuses on the modification of the flows and exchanges in the air gap between two discs using an impacting air jet.

The convective transfer of heat in a rotating disk has been studied by [2] who proposes various correlations the local and average Nusselt numbers in the event of the radial temperate profile on the surface of the disk. [3] showed that the heat transfer is significant near the point of impact of the jet on the disk. [4] experimentally studied the heat transfers for the case of the unconfined rotor/stator configuration in the face of an axi-symmetric air jets for different scale spacings $(10^{-2} < G < 16^{-2})$ for Re_j ranging from 0 to 41666 and for Re_ω ranging from 20000 to 516000. They illustrated that the spinning disk surface can be separated into three areas where the effect of the jet or spin on the heat transfer is dominant. The three areas do not necessarily have to occur on the disk face at the same time. The three zones do not have to be present at the same time on the surface of the disc. They are dependent on the injected rate of flow and the speed of rotation, among other things. They suggested correlations of mean Nusselt numbers for various values of the spacing G for $(0.01 < G < 0.04)$ $\overline{Nu} = 0.08 Re_j^{-0.07} Re_\omega^{0.25}$ [4] and for $(0.04 < G < 0.08)$ $\overline{Nu} = 0.006 G^{0.15} Re_j^{0.5} Re_\omega^{0.5}$ [4].

Regardless of the value of the spacing G, the author [4] showed that there is a single peak on the local Nusselt number profile near the stagnation point. The [5] experimentally studied the flow characteristics of an air jet in rotor/stator geometry consisting of a small non-dimensional spacing. Their experimental study indicated the existence of a centrifugal type of flow near the impact region. Measurements show that rotation has a limited effect on the flow near the impact point. For larger radii, rotation dominates and the flow becomes centrifugal. [6] have completed the experimental study of [5]. They have compared the experimental results obtained by PIV ("Particle Image Velocimetry") with turbulence models $K\_\omega SST$ and RSM. The turbulence models allowed to find the different characteristic zones of an impacting jet with heat transfer. The $K\_\omega SST$ model shows a tendency to over-predict the flow properties in the vicinity of the impact. The results of the Reynolds stress model are similar with the experimental results.

On the other hand [6], et al. revealed the presence of two peaks. [7] reported that the first peak is at $r/(2Re_j) = 1/2$ and the second is provided by the formula $(r/2Re_j) = 0.188 Re_\omega^{0.241} (H/2Re_j)^{0.224}$ [7].

Similarly, direct numerical simulations (DNS) have been performed by [8]. The geometric properties of the system in [8] are consistent with the experimental setup develop the experimental setup by [4]. The jet relative Reynolds number is fixed at $Re_j = 5300$ while the rotating Reynolds number Re_ω can vary to conserve the vortex parameter N between $0 \le N = \frac{Re_\omega}{Re_j} \times \frac{D}{R_d} \le 2.47$ [8].

The Nusselt number distribution, characterizing the heat transfers, reveals the presence of two summits. The first summit is due to high fluid velocities inducing large shear stresses and the second peak is related to the boundary layer transition. Recently [9] the cooling of a rotating disk by an eccentric jet impacting on a rotor-stator cavity was studied. The geometry properties of the system correspond to the experimental setup developed by [4], The rotational Reynolds numbers of Re vary from 2.83×10^5 to 5.44×10^5 and the Reynolds number of the jet varies from 16.5×10^3 to 49.5×10^3. The numerical study in [9] based on RSM (Reynolds Stress Model) turbulence. The results of this work are described as radial velocity and average Nusselt numbers. For a jet Reynolds number set at $Re_j = 33 \times 10^3$, the local Nusselt numbers show two peaks located simultaneously at $r/D = 5$ and $r/D = 6.5$.

2. NUMERICAL SIMULATION

2.1 Geometrical Model and Boundary Conditions

The rotor-stator cavity shown in figure 1 is modeled by two equal radius disks $(R = 310\,\text{mm})$ one rotating (rotor) and the other fixed (stator). The air flows through a cylindrical inlet tube before impacting the rotating disk. The dimensioned distance separating the two discs is varied between $G = 0.02$ and $G = 0.04$. The air passes through a cylindrical tube of diameter $D = 26\,\text{mm}$ and length $750\,\text{mm}$ before impacting on the rotating disc. The aerodynamic fluid flow and heat transfer in forced convection are mainly impacted by the jet-related and rotational Reynolds numbers defined as: $Re_\omega = \frac{\omega \times R_d^2}{\nu}$ and $Re_j = \frac{U_j \times D}{\nu}$, where ν is the kinematic viscosity of the air.

2.2. Meshing

A tetra-frame mesh figure 2 has been selected as it is adapted to the chosen turbulence model. The short distance separating the two disks imposes a very fine mesh [11].

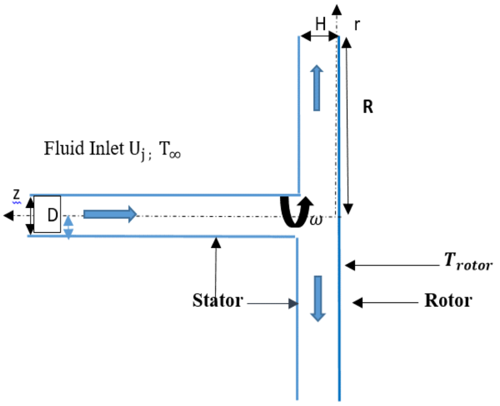

Figure 1. Geometry of the rotor-stator system

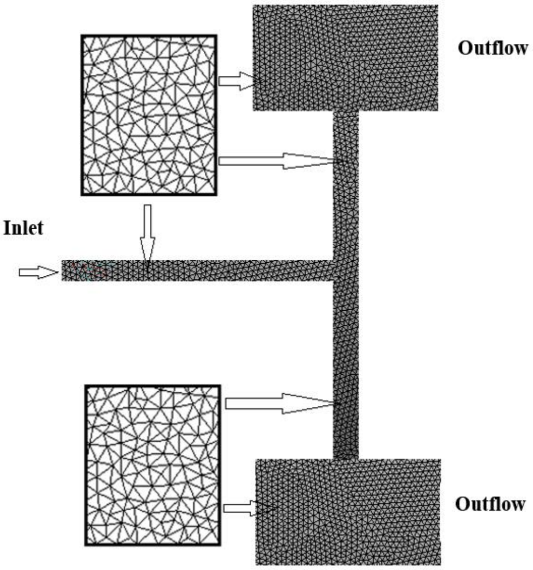

Figure 1. Overview of the type of mesh used

Different types of boundary conditions:

- At the fluid input, a constant velocity and temperature $(U_j$ and $T_\infty)$ are impose. An imposed turbulence rate corresponds at a hydraulic diameter of $26\,\text{mm}$ and a turbulent intensities of 5%. On the walls, $U_r = U_z = 0$, except the tangential velocity is equal to $U_\theta = \omega r$ on the rotating walls and $U_\theta = 0$ on the fixed walls.
- The flowing fluid is air with Prandtl number $(P_r = 0.72)$ that enters the cavity with a temperature $T_\infty = 25\,^\circ\text{C}$, which is the same at the stator interface.
- The Temperature on the right side of the rotor is is $T_r = 80\,^\circ\text{C}$.

2.3 Turbulence Model and Numerical Scheme

ANSYS Fluent [15] has different models to characterize the turbulent nature of flows. The RSM (Reynolds Stress Model), allows a detailed description of the turbulent field in the whole domain, including the turbulent fields near the walls and has the advantage of not introducing the viscosity assumption of the turbulence.

The Equation of elements of the Reynolds tensor R_{ij} is:

$$\overline{u_j}\,\frac{\partial \overline{u_i' u_j'}}{\partial x_j} = P_{ij} + \Omega_{ij} + \Phi_{ij} + D_{ij} - \varepsilon_{ij} + O_{ij} \qquad (1)$$

where, $P_{ij}, \Omega_{ij}, \Phi_{ij}, D_{ij}, \varepsilon_{ij}, O_{ij}$ simultaneously represent: production, diffusion, pressure-strain correlation, dissipation and implicit effect of rotation on turbulence.

The energy equation and the transport equations are solved by SIMPLE scheme. The second order spatial transposition is selected for the solution of the convective coefficients; we have opted for the PRESTO scheme for the pressure correction equations.

3. RESULTS AND DISCUSSION

3.1. Streamlines

The impacting air jet creates a radial flow, this flow is enclosed in a recirculation zone as shown in Figure 3. The size of the recirculation zone is not dependent on rotational Reynolds number and seems similar to the results of [8] and [6] which show that the recirculation zone size varies as a function of G-spacing.

For $\frac{r}{R_d} > 0.16$ beyond the recirculation zone the flow becomes centrifugal. The streamlines are parallel to the rotor which is similar to the results obtained by [8]. For $Re_\omega = 3.35\times10^5$ and $Re_\omega = 4.69\times10^5$ which correspond to very high rotational speeds, the flow away from the impact zone is rotationally dominated. A large recirculation zone is created near the stator. For very high rotational speeds, the flow is centrifugal near the rotor and a reentry of fluid from the outside is observed near the stator. This re-entry of fluid from the outside was also observed by [8] who showed that for $N = 1.237$ the flow becomes tangential from $\frac{r}{R_d} = 0.61$ and for $N = 2.47$ the flow becomes tangential from $\frac{r}{R_d} = 0.45$. The [6] and [5] have shown that the flow becomes tangential from $\frac{r}{R_d} = 0.6$ and $\frac{r}{R_d} = 0.4$. The increase in rotational speed results in a transition from centrifugal to tangential flow.

3.2. Flow Structures

For various values of Re_ω (rotational Reynolds number) and for Re_j fixed Figure 4 represents the average speeds of air in the air gap. Figure 4a illustrates that the rotational speed of the rotor has no effect on the axial velocity of the fluid $(\bar{U}_z / U_j)$ since the jet-related Reynolds number is constant and the rotation has no effect on the impact zone, the field $(\bar{U}_z)$ undergoes no change when Re_ω increases. In contrast, the influence of Re_ω is apparent on the radial velocities as shown in Figure 4b.

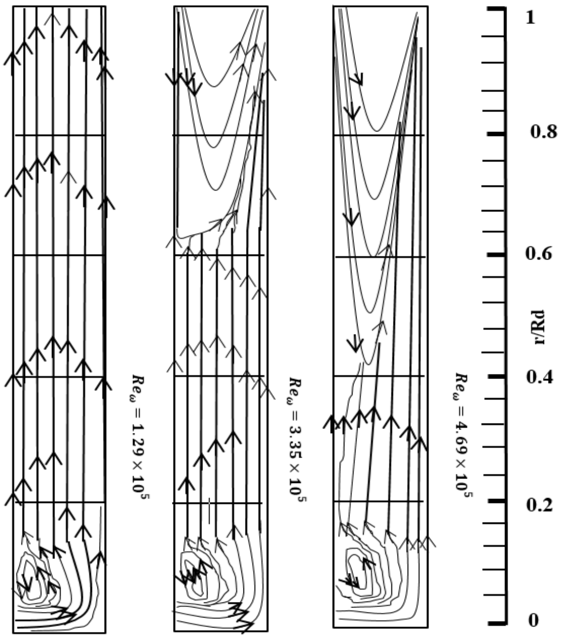

Figure 2. Current lines obtained for $G = 0.04$ and $Re_j = 16.6\times10^3$

The negative radial velocities characterize the recirculation zone that correspond to a fluid reentry from the outside. This inflow of fluid has also been observed by [6] and [5]. The air flow in the vacuum part is centripetal on the stator and centrifugal on the rotor. Near the impact zone, the radial velocity field $(\bar{U}_r / U_j)$ shows negative over speeds near the stator and positive over speeds near the rotor.

These over speeds indicate the change of direction of the flow and its confinement by the area of recirculation. The Figure 4c illustrates that the flow is rotating quasi-solid. The flow is progressively dominated by the rotor and the boundary layer. The flow is progressively dominated by the spin.

The tangential and radial velocity profiles of the fluid give an idea about the flow structures. A flow of [12] is a flow composed of two distinct boundary layers separated by a core of rotating non-viscous fluid. Another flow with only one boundary layer and almost zero tangential velocity is called a [13] flow.

In Figure 5, we present the tangential and radial velocity profiles of the fluid gives an idea on the flow structures. The numerical simulation is done for different values of Re_ω and for fixed Re_j.

For $Re_j = 16.6\times10^3$ and for $Re_\omega = 4.69\times10^5$ (Figure 5a), the radial components of the velocity obtained for different radial positions represent a parabolic profile. The maximum value of this component is obtained for $Z/H = 0.5$. The values of the tangential component obtained for different radii are very important near the rotor $(Z/H = 0)$, where the rotation of the disk is the main driver of the flow.

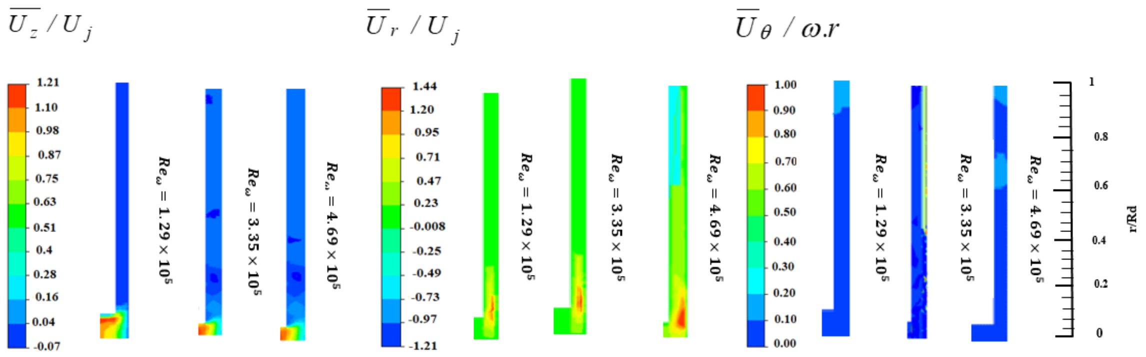

Figure 4. Average speeds obtained by digital simulation; (a) $(\bar{U}_z / U_j)$; (b) $(\bar{U}_r / U_j)$; (c) $(U_\theta / \omega r)$; for $Re_j = 16.6 \times 10^3$

They decrease more when moving away from the rotor and they become zero for $(Z/H = 1)$. The tangential velocities have only one boundary layer near the rotor, moving away from this boundary layer, the tangential velocities decrease and tends to zero when (Z/H) will be close to 1. The obtained flows are similar to flows of [13].

In Figures 5b and 5c, we can see that near the stator, and for values of $\frac{r}{R_d} = 0.55$ and $\frac{r}{R_d} = 0.8$, the radial velocities decrease as a function of Z and becomes negative. This corresponds to an entry of air from the outside into the air gap near the stator. However, we distinguish two areas: one where the tangential velocities are negative which indicates that the flow is centrifugal and other positive radial velocities that reveal that the flow is centripetal. all this characterizes flow of Batchelor [12].

3.3. Nusselt Number

Figure 6 presents the results of local Nusselt number (Nur) on the surface of the left rotor as a function of scaled radius (r/D), for $G = 0.02$ and $Re_\omega = 1.29 \times 10^5$ for different values of jet related Reynolds number ranging from $Re_j = 16.6 \times 10^3$ to $Re_j = 38.133 \times 10^3$. The influence of air flow variation injected by the center of the stator is manifested by an augmentation of the local Nusselt number (Nur) near the impact zone of the jet. The authors [14], [4] and [8] have noticed that in the presence of an air jet, the convective thermal energy increases and the more one moves away from jet the influence of the jet decreases.

The numerical results provided by the RSM turbulence model are in agreement to the experimental results [4]. The average related variations obtained remain within the margin of error of the experimental measures with a variation of less than 5%. Given the accuracy of the experimental results, the CFD model can be concluded to accurately predict the behavior of convective exchanges on the left side of the rotor. For $G = 0.04$ and Re_j fixed Figure 7 shows that there is a zone near the center of the left rotor, where the convective exchanges are not influenced by the rotation speed, this is in agreement with the observations of [9, 10].

The size of the impact zone depends on the dimensional space G, it increases as G increases. At the periphery of the left rotor, there is a zone where the exchanges are not influenced by Re_j.

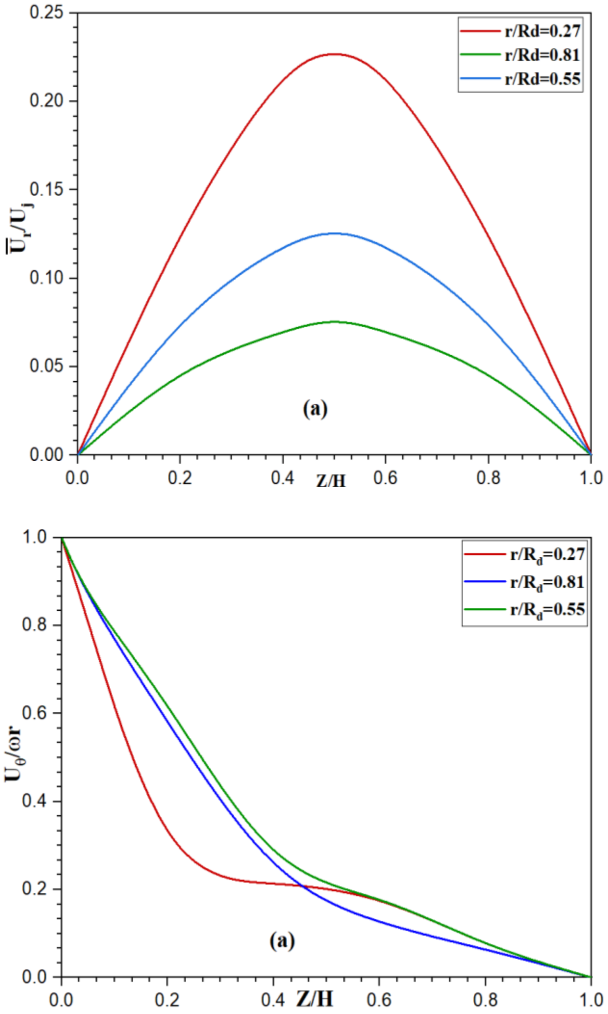

Figure 5. Axial distribution of radial velocity $(\bar{U}_r / U_j)$ and tangential velocity $(U_\theta / \omega r)$ for $G = 0.02$ and $Re_j = 16.6 \times 10^3$, Three positions are considered: ($r/R_d = 0.27$ $r/R_d = 0.55$ $r/R_d = 0.81$), for three rotational Reynolds numbers $Re_\omega = 1.29 \times 10^5$ (a) $Re_\omega = 3.35 \times 10^5$ (b) and $Re_\omega = 4.69 \times 10^5$ (c)

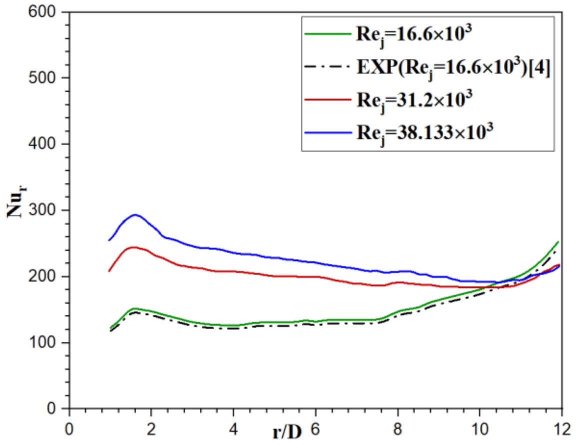

Figure 6. Local Nusselt Number (Nur), $G=0.02$ and $Re_\omega=1.29\times10^5$

Figure 8 illustrates the evolution of the average Nusselt number as a result of the rotational Reynolds number for different values of Re_j. On this same Figure, we have represented the results obtained by [17] in the example of the rotor-stator system not using jets and the results obtained by [14] concerning a rotating disk placed in front of a stator with a central opening in the stator. The numerical results of the average Nusselt number are higher than those provided by [16] and [17]. In effect, the existence of an injected air flux leads to an air renewal in the air gap and the cooling gets more important.

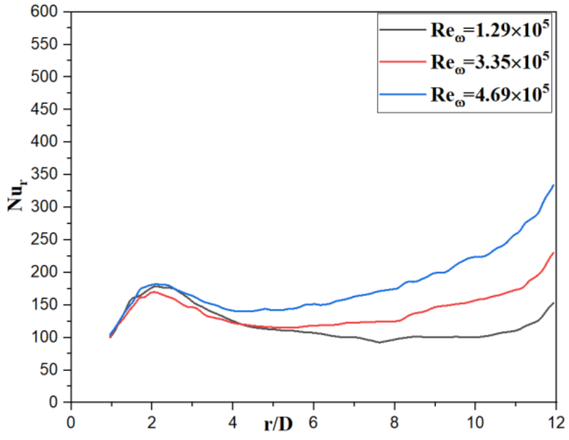

Figure 7. Local Nusselt Numbers for $G=0.04$ and $Re_j=16.6\times10^3$

The increase of the injected air flow results in an augmentation of the convective transfers at the rotor surface. On the other hand, the comparison of our numerical results with the experimental data [4] obtained in the example of the rotor-stator system using a jet and of a fixed Reynolds number, shows a satisfactory agreement with the use of the RSM model of turbulence.

A correlation made by [4] to relate this average Nusselt number $\overline{Nu}$ to the rotational Reynolds number Re_ω and the jet-related Reynolds number Re_j which is consistent to the numerical result. For $0.01<G<0.02$ the laws found by [17] are $\overline{Nu}=0.08G^{-0.07}Re_j^{0.5}Re_\omega^{0.25}$.

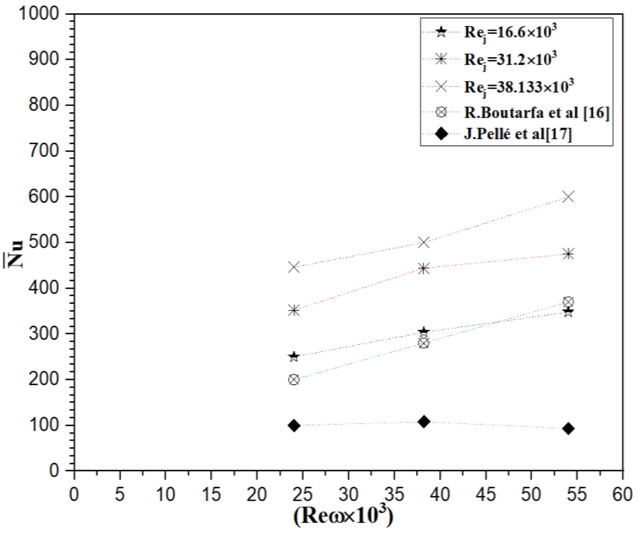

Figure 8. Comparison of the numerical results obtained in this study with [16] and [17] for $G=0.02$

4. CONCLUSIONS

The flow characteristics of a jet impacting a rotating disk subjected to a constant temperature have been studied using the Ansys-Fluent calculation code. Two different zones have been identified; a recirculation zone close to the impact zone and characterized by strong heat transfers and a zone at the periphery of the cavity where the effects of the rotation dominate and where the heat transfers are strongly dependent on them. Heat transfer depends on jet-related Reynolds number ranging from 16.6×10^3 to 38.133×10^3 and rotational number, ranging in our study from 1.29×10^5 to 4.69×10^5. It also depends on the spacing G. The obtained numerical results are in good agreement with the experimental data of [3] with a deviation of less than 5%. An analysis of the profiles of the radial and tangential components of the air velocity in the space between the two discs shows that the structure of the flow is close to that of [10] with a centrifugal flow located in the vicinity of the rotor and another centripetal flow located in the vicinity of the stator.

REFERENCES

[1] F. Cabrera Quintero, J.F. Medina Padron, E.J. Medina Dominguez, M.A. Artiles Santana, "Renewable Hydro-Wind Power System for Small Islands: The El Hierro Case", International Journal on Technical and Physical Problems of Engineering (IJTPE), Issue 27, Vol. 8, No. 2, pp. 1-7, June 2016.

[2] L. Dorfman, "Hydrodynamic Resistance and the Heat Loss of Rotating Solids", Oliver and Boyd, Vol. 100, 1963.

[3] M. Angioletti, R.M. Di Tommaso, E. Nino, G. Ruocco, "Simultaneous Visualization of Flow Field and Evaluation of Local Heat Transfer by Transitional Impinging Jets", Int. J. Heat Mass Transf, Vol. 46, pp. 1703-1713, 2003.

[4] 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.

[5] 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.

[6] 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.
[7] D. Lytle, W. Webb, “Air Jet Impingement Heat Transfer at Low Nozzle-Plate Spacings”, Int. J. Heat Mass Transfer, Vol. 37, pp. 1687-1697, 1994.
[8] 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, Issue 61, pp. 366-378, 2016.
[9] 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, September 2019.
[10] C.O. Popiel, L. Boguslawski, “Local Heat Transfer from a Rotating Disk in an Impinging Round Jet”, ASME J. Heat Transfer, Vol. 37, pp. 357-364, 1986.
[11] H. Yaagoubi, H. Abouchadi, M.T. Janan, “3d Mathematical Thermal Analyses of PA12 Powder Bed on the SLS Process by Two Numerical Methods”, International Journal on Technical and Physical Problems of Engineering (IJTPE), Issue 51, Vol. 14, No. 2, pp. 74-83, June 2022.
[12] 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.
[13] 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, UK, 1953.
[14] J. Pelle, S. Harmand, “Heat Transfer Measurements in an Opened Rotor-Stator System Air-Gap”, Experimental Thermal and Fluid Science, Vol. 31, pp. 165-180, 2007.
[15] A. Fluent, “12.0 Theory Guide”, Ansys Inc, Vol. 5, No. 5, 2009.
[16] R. Boutarfa, S. Harmand, “Local Convective Heat Transfer for Laminar and Turbulent Flow in a Rotor-Stator System”, Exp. Fluids, Vol. 38, No. 2, pp. 209-221, 2005.
[17] J. Pelle, “Experimental Study of Convective Exchanges on the Rotor of a Discoidal Machine: Influence of an Impacting Jet”, Valenciennes, 2006.

BIOGRAPHIES

Abdellatif El Hannaoui was born in El Brouj, Morocco on January 9, 1989. He obtained a state engineering degree in industrial engineering from Faculty of Technical Sciences, Hassan I University, Settat, Morocco in 2014. He is currently a Ph.D. student in research team of Industrial Management Engineering and Innovation at the same university. His research interests include the study of heat transfer in a discoid machine.

Chadia Haidar was born in Fkih Ben Salah, Morocco on March 3, 1991. She earned the engineering degree in processes engineering energy and environment from ENSA Khouribga, Morocco in 2014. She received the Ph.D. degree in mechanics and energy from Faculty of Sciences and Technologies, Hassan I University, Settat, Morocco. Her thesis work focuses on the numerical and experimental analysis of thermal transfers on the rotor of a discoidal machine.

Rachid Boutarfa was born in Benselimane, Morocco on January 18, 1973. He received the Ph.D. degree in Energetic from University of Valenciennes and Hainaut-Cambresis, Valenciennes, France in 2001. He is a Professor at Mechanical Engineering Department, Faculty of Sciences and Technologies, Hassan I University, Settat, Morocco since 2004. His research activities are related to numerical and experimental study of convective heat transfer on rotating machines in order to improve the energy efficiency of thermal systems.