Comparative Investigation on Heated Swirling Jets Using Experimental and Numerical Computations

Ali Khelil\textsuperscript{a}, Hassane Naji\textsuperscript{b,c}, Mohamed Braikia\textsuperscript{a} & Larbi Loukarfi\textsuperscript{a}

\textsuperscript{a} Université de Chlef, Chlef, Algérie
\textsuperscript{b} Université d’Artois, Laboratoire Génie Civil et géo-Environnement, Béthune, France
\textsuperscript{c} Université Lille Nord de France, Lille, France

Accepted author version posted online: 25 Mar 2014. Published online: 08 Aug 2014.


To link to this article: http://dx.doi.org/10.1080/01457632.2014.906279

PLEASE SCROLL DOWN FOR ARTICLE

Taylor & Francis makes every effort to ensure the accuracy of all the information (the “Content”) contained in the publications on our platform. However, Taylor & Francis, our agents, and our licensors make no representations or warranties whatsoever as to the accuracy, completeness, or suitability for any purpose of the Content. Any opinions and views expressed in this publication are the opinions and views of the authors, and are not the views of or endorsed by Taylor & Francis. The accuracy of the Content should not be relied upon and should be independently verified with primary sources of information. Taylor and Francis shall not be liable for any losses, actions, claims, proceedings, demands, costs, expenses, damages, and other liabilities whatsoever or howsoever caused arising directly or indirectly in connection with, in relation to or arising out of the use of the Content.

This article may be used for research, teaching, and private study purposes. Any substantial or systematic reproduction, redistribution, reselling, loan, sub-licensing, systematic supply, or distribution in any form to anyone is expressly forbidden. Terms & Conditions of access and use can be found at http://www.tandfonline.com/page/terms-and-conditions
Comparative Investigation on Heated Swirling Jets Using Experimental and Numerical Computations

ALI KHELIL,¹ HASSANE NAJI,²,³ MOHAMED BRAIKIA,¹ and LARBI LOUKARFI¹
¹Université de Chlef, Chlef, Algérie
²Université d’Artois, Laboratoire Génie Civil et géo-Environnement, Béthune, France
³Université Lille Nord de France, Lille, France

The purpose of this paper is to investigate both experimentally and numerically the influence of various parameters on the different blowing configurations of multiple swirling jets. Flow rate was adjusted at Reynolds numbers ranging from $10^4$ to $3 \times 10^4$. The current study is carried out under uniform heat flux condition for each diffuser, at Reynolds number of $3 \times 10^4$, with air being the working fluid. Experiments concerning the fusion of several jets show that the resulting jet is clearly more homogenized under swirling influence. Afterward, numerical simulation is also carried out using the finite-volume computational fluid dynamics solver FLUENT 6.3, in which the standard $k-\varepsilon$ and the Reynolds stress turbulence model (RSM) were used for turbulence computations. The findings of this study show that the diffuser vane angle and a balance and an imbalance in temperature between the central and peripheral jets affect the quality of the thermal homogenization of the ambiance. Overall predictions obtained with the RSM model are in better agreement with the experimental data compared to those of the standard $k-\varepsilon$ model.

INTRODUCTION

Swirling jets are widely encountered in engineering facilities, such as for cyclone combustors, combustion engines, tangentially fired furnaces, swirl burners, cleaning, and cooling and heating, among others. They are frequently used to improve heat transfer in heating and ventilation. The azimuthal motion can be given to the jet by different mechanisms, for example, by using inclined vanes. The understanding of swirl effects is very important for the efficiency of the ventilation process. However, to our knowledge, these effects have been scarcely investigated, and consequently the fusion of many swirling jets becomes interesting to study. The multiple swirling free jets studies show that swirling jets will develop more rapidly than jets without swirl. Note that the number of jets contributes to decrease velocities. Also, the distance between blowing orifices involves a decrease of velocities while delaying jets fusion. For high swirl numbers and far from the orifices, velocity profiles of multiple jets have a tendency to increase compared with those of the single jet. The interaction between jets allows the distribution of velocities in the mixing zone in which the normal stresses and maximum shear are located. Near the origin of the swirler, the profiles are characterized by irregularities due to the swirler geometry and the blowing conditions [1]. It should be noted that the axial temperature for the multijet seems to have an exponential decrease [2–10]. According to the available literature, there is no advanced research being done on the multiple swirling jets applied to improve thermal homogenization. Most papers that deal with multiple swirling jets in various geometric, dynamic, and thermal conditions are aimed at the improvement of combustion [11]. Yimer et al. [12] experimentally studied the development of the flow from multiple-jet cold-model burners. They measured the fields of mean velocity and velocity fluctuation intensity with Pitot probes. They noted that beyond the near field, flows are virtually the same and are similar to those of a round jet from a single source, and in the near field,
individual structures peculiar to each burner are observed. Yin [13] experimentally investigated twin jets flow, generated by two identical parallel axisymmetric nozzles. He found that the twin jets attract each other. With increasing Reynolds number, the turbulence energy grows, which indicates that the twin jets attract acutely. He also observed that the jet flow field and the merging process of twin jets vary with the spacing between two nozzles. One of his salient findings is that the width of the twin jets flow spreads linearly downstream and grows with the spacing between two nozzles.

The effect of various parameters on flow development behind a vane swirler was extensively studied experimentally and numerically by Raj and Ganesan [14]. This work highlighted the main features of flow field generated by vane swirler. The uniqueness of this study is in arriving at the best vane angle using appropriate turbulence models for both weak and strong swirl. It is found that for a weak swirl, the standard $k−\varepsilon$ model is sufficient, whereas for strong swirl one has to resort to the Reynolds stress model (RSM). They found that the latter model becomes more appropriate as the swirl is increased. As mentioned before, previous studies showed that for a range of experimental conditions, heat transfer enhancement is strongly dependent on blade angle, and it seems that the Reynolds number $Re$ has no significant influence, whereas when the initial angle of the velocity $\alpha$ increases, the jet is more spreading in the radial direction [15].

Ahmadvand et al. [15] studied experimentally and numerically the influence of the axial vane swirler on increase of heat transfer and turbulent fluid flow. Their study has been carried out for three blade angles of 30°, 45°, and 60° with uniform heat flux condition of air, which is used as the working fluid. These authors confirmed that the use of a vane swirler leads to a higher heat transfer compared with those obtained from plain tubes, and the thermal performance increases as vane angle is raised and decreases by growth of Reynolds number. Wang et al. [16] conducted a numerical study indicating that multiplet injection, especially four jets spaced circumferentially 90° apart, was a favorable configuration to enhance mixing as compared with single-jet injection, and that an even number of opposing jets may be preferable to an odd number of opposing jets. Later, Giorges et al. [17] carried out a systematic numerical study for single- and multiple-jet injections into a main stream using the standard $k−\varepsilon$ turbulence model available in the computational fluid dynamics (CFD) code FLUENT. Their numerical results displayed that multiple opposing jet injection not only gave better mixing but decreased power requirements when the number of side inlet jets increased. Suyambazhahan et al. [18] studied numerically non-isothermal twin parallel jets in horizontal orientation to ascertain the main flow structure and the oscillation characteristics of temperature and velocity fields. Such analysis is carried out for Reynolds number between $9 \times 10^3$ and $12 \times 10^3$ using the standard $k−\varepsilon$ model. They found that the simulation results compare well with available experimental data on axial velocity distribution and jet merger distance. Wang and Mujumdar [19] studied the fluid flow and mixing characteristics of multiple and multiset three-dimensional confined turbulent round opposing jets in a novel in-line mixer using the standard $k−\varepsilon$ turbulence model. They obtained a good agreement between the simulated results and experiments.

From the literature review, it appears that vortex flows undeniably have some advantages in terms of power mixing. As mentioned already, all research studies conducted on such kinds of flow are somewhat remote from our present study. Thus, assessing the relevance of integrating the turbulent jets in the air handling and ventilation of living spaces and transport requires prior study and analysis of multijet swirling over its entire length. For that reason, the choice of a system consisting of blow jets more efficient in terms of mixing may be necessary. More recently, Escue and Cui [20] presented a numerical study of the swirling flow inside a straight pipe. Computations are performed using FLUENT commercial software [21]. The turbulent models used in that study include the $RNG − k−\varepsilon$ and the Reynolds stress model (RSM). They found that the latter model becomes more appropriate as the swirl is increased. As mentioned before, previous studies showed that for a range of experimental conditions, heat transfer enhancement is strongly dependent on blade angle, and it seems that the Reynolds number $Re$ has no significant influence, whereas when the initial angle of the velocity $\alpha$ increases, the jet is more spreading in the radial direction [15]. In this study, the dimensionless temperature and $r$- and $x$-coordinates were normalized in the form $T_r = (T − T_a) / (T_0 − T_a)$, $r/D$, and $x/D$, respectively, where $T$ is the jet temperature, $T_a$ is the ambient temperature, and $T_0$ is the maximum temperature of the air blowing at origin.

From the preceding discussion, the main aim of this study is to examine the influence of various parameters such as the number, the arrangements of the single or multiple swirling jets, and the diffuser vane angle on the flow resulting both dynamically and thermally. In addition, we expect that parametric study of these parameters will help to optimize the choice of the configuration of interest to industry and the choice of the turbulence model that adapts with the numerical simulation of the multiple swirling jets.

**EXPERIMENTAL SETUP AND TECHNIQUES**

The experimental facility is depicted in Figure 1. It consists of a size chassis $2000 \times 800 \times 400$ (mm), which is fixed on a square plate of Plexiglas. On the latter, three devices blowing hot air (hairdryer TEFAL-1500) are fixed and directed downward, and the lower part of these devices is used to fix different types of diffusers provided with inclined vanes, depending on the studied configuration. Temperatures and velocity of the flow are measured by a hot-wire anemometer (type Velocicalc Plus Air Velocity Meter [22]), which is a high-precision multifunctional instrument. The data can be viewed on a screen, printed, or downloaded to a spreadsheet program, allowing us to easily transfer data to a computer for statistical treatment. The accuracy is of order $\pm 0.015 \text{ m/s}$ for velocity and $\pm 0.1^\circ \text{C}$ for temperature from the thermal sensor. Note that the thermal sensor is supported by rods that are easily guided vertically and horizontally to sweep the maximum space in the axial and radial directions.
The swirling free jet considered here is different from the conventional jet because of the existence of a tangential component velocity. To obtain this kind of flow, one can use swirling mechanical systems. For example, this system includes inclined vanes (see Figures 2a–2c), which are put in the generating tube jet (see Figure 2b). The application of a tangential velocity component to the flow \( W \) provides a rotation to flow fluid, which is indicated by a so-called swirl number \( S \). This number is defined as the ratio of the axial flux of tangential fluid, which is indicated by a so-called swirl number \( S \), to the product of the axial momentum flux and a characteristic radius \( R \).

\[
S = \frac{G_\theta}{RG_x} = \frac{\int_{R_n}^{R_h} UW r^2 dr}{\int_{R_n}^{R_h} R_n U^2 rdr} \tag{1}
\]

where \( G_\theta \) is the axial flux of tangential momentum, \( G_x \) is the axial momentum flux, and \( R \) is a characteristic radius. \( R_n \) and \( R_h \) are radius of the center body and the inlet duct, respectively. It is important to note here that if the axial and azimuthally velocities are assumed to be uniform and the vane are very thin, the swirl number can be expressed as

\[
S = \frac{2}{3} \left[ \left(1 - \left(\frac{R_h}{R_n}\right)^3\right) / \left(1 - \left(\frac{R_h}{R_n}\right)^3\right) \right] \tan \alpha \tag{2}
\]

where \( \alpha \) is the swirler vane angle. Throughout the remainder of this paper, this parameter is called the vane angle.

In this study, the axial and tangential velocities \( U \) and \( W \) were measured at the exit of a swirling jet diffuser with a triple-probe hot wire anemometer (DISA 55M01). Four swirl numbers values are considered in this study. These are \( S = 0 \) for \( \alpha = 0^\circ \), \( S = 0.4 \) for \( \alpha = 30^\circ \), \( S = 0.7 \) for \( \alpha = 45^\circ \), and \( S = 1.3 \) for \( \alpha = 60^\circ \), respectively.

To carry out our experiments, the following operating conditions have been considered: \( 0 < S < 1.3 \), \( R_e_0 = 30 \times 10^3 \), \( r/D = 1 \) to \( 8 \), and \( 0 \leq x/D \leq 20 \). Here, it is useful to note that previous studies were based on similar ranges of Reynolds number (see Sislian and Cursworth [24] and Volchkov et al. [25], among others). It is worth recalling that here our goal is to identify and study the evolution of temperature profiles of axial and radial multijets swirling in different configurations. This approach allows analyzing the influence of key parameters such as angle of inclination of the vanes \( \alpha \) and the number of peripheral jets, which are controlled by a central jet. Two configurations consist of three swirling jets in imbalance (A) and in balance (B) with temperature and a single swirling jet configuration (C) are presented in Figure 3.

**NUMERICAL SIMULATION PROCEDURE**

**Mathematical Modeling**

For a steady, three-dimensional, incompressible, and turbulent flow with constant fluid properties, the governing equations of conservation of mass, momentum, and energy are written in the Cartesian tensor notation as follows:

\[
\frac{\partial U_i}{\partial x_i} = 0 \tag{4}
\]

\[
\frac{\partial (U_i U_j)}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \rho \theta u_i u_j \right] \tag{5}
\]
\[ \rho C_p U_i \frac{\partial T}{\partial x_i} = \frac{\partial}{\partial x_i} \left[ k \frac{\partial T}{\partial x_i} - \rho C_p u_i' T' \right] \]  

where \( U_i' \) and \( T' \) denote the mean velocity and temperature; \( u_i' \) and \( T' \) are the corresponding fluctuation components; and \( -\rho u_i' u_j' \) and \( -\rho C_p u_i' T' \) are the average Reynolds stresses and turbulent heat fluxes, which need to be modeled to close the equations. It should be noted that here, the temperature variations are negligible and the Mach number is low (\(<0.3\)), which allows us to assume that the fluid is incompressible (constant density).

The Boussinesq hypothesis, which relates the Reynolds stresses to the mean velocity gradients, is expressed as

\[ -\rho u_i' u_j' = \mu_t \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} \left( \rho k + \mu_t \frac{\partial U_i}{\partial x_i} \right) \delta_{ij} \]  

where \( k \) is the turbulent kinetic energy, as defined by \( k = \frac{u_i' u_i'}{2} \), and \( \delta_{ij} \) is the tensor identity. An advantage of the Boussinesq approach is the relatively low computational cost associated with the computation of the turbulent viscosity \( \mu_t \). Note that the turbulent viscosity \( \mu_t \) is given by

\[ \mu_t = C_\mu \rho k^2 / \varepsilon \]  

with \( \varepsilon \) being the turbulence energy dissipation rate, and \( C_\mu \) is a turbulence modeling constant.

Hence in the present calculation, the turbulence scalar fluxes are modeled using the gradient-diffusion approach [26] as

\[ \rho u_i' T' = -\frac{\mu_t}{\sigma_t} \frac{\partial T}{\partial x_i} \]  

where \( \sigma_t = 0.6 \) stands for the turbulent Prandtl number.

The \( k-\varepsilon \) model is an example of two equation models that use the Boussinesq hypothesis. Here, two different closure models, the \( k-\varepsilon \) model and the Reynolds stress model (RSM), are used.

**RSM Model**

Two-equation turbulence models (\( k-\varepsilon \) or others) offer good predictions of the characteristics and physics of most flows of industrial relevance. In flows where the turbulent transport or nonequilibrium effects are important, the eddy-viscosity assumption is no longer valid and results of eddy-viscosity models might be inaccurate. Reynolds tress models have shown superior predictive performance compared to eddy-viscosity models due to their isotropic nature. Therefore, anisotropic models such as the full Reynolds stress transport models (RSM) are necessary for accurate prediction of turbulent swirling flows. In RSM, the eddy viscosity approach has been discarded and the Reynolds stresses are directly computed.

The transport equations for the Reynolds stresses \( -\rho u_i' u_j' \) are expressed in a general form (neglecting the effects of buoyancy) as

\[ \frac{\partial}{\partial x_k} \left( \rho U_k u_i' u_j' \right) - \frac{\partial}{\partial x_k} \left( \mu_t \frac{\partial u_i' u_j'}{\partial x_k} \right) = D_{T,i,j} + P_{ij} + \varphi_{ij} \]  

where

- \( D_{T,i,j} \) represents the turbulent diffusion,
- \( P_{ij} \) represents the production,
- \( \varphi_{ij} \) represents the pressure-strain term.
where the various terms from left to right represent, respectively, convection, diffusion, production, pressure–strain redistribution, viscous dissipation, and additional production of the stresses. Note that the convection and production terms are exact, while the remaining terms have to be modeled [27]. Detailed derivations for the closure equations can be found in references [28–32], among others.

**Grid Generation**

It goes without saying that the strategy of grid generation within the computational region and the density of the grid play an important role in the prediction accuracy. The grid was nonuniform, with high density in zones of great interest and low density in zones of less interest, so that minimal computational effort was required while gaining sufficient accuracy. The computational grid geometry of the swirl generator and the entire inflow system are presented in Figures 4 and 5, respectively. Tetrahedral mesh has been used, and in order to capture wall gradient effects, mesh has been finer toward the vanes. In the radial direction the mesh is fine over the test inlet and then stretched to the exit. Tests with finer grids (up to 18 × 10^5 cells) demonstrate that the quality of the prediction is not improved...
by enhancing the number of cells used (see Table 1). Computations on different mesh show that the solution of the radial dimensionless temperature in the case of configuration C does not change significantly (errors remain on the order of ≤4%), which suggests that the solution is independent of the mesh, as can be seen in Figure 6.

Note that there are a total of 1,839,831, 2,244,126, and 973,556 cells for blowing configurations A, B, and C, respectively.

**Boundary Conditions**

The following boundary conditions were specified in all cases simulated: at the inlet of the blowing configuration, uniform axial velocity (6 m/s) and maximal temperature $T_0 = 90^\circ C$ (= 363 K) and $T_0/2 = 45^\circ C$ (= 318 K); the initial turbulence intensity, $I$, also often referred to as turbulence level, is estimated from the following empirical correlation:

$$ I = 0.16(\text{Re}_D)^{-1/8} $$  

where $I$ is given as a percentage (see Table 2), and $\text{Re}_D$ is the Reynolds number based on the inner diameter $D$ of the diffuser.

It is worth recalling that an approximate relationship between turbulence length scale $l$ and the physical size of the flow diffuser diameter $D$ was used for turbulence energy modeling (see [28]):

$$ l = 0.07D $$  

**Numerical Predictions**

In this work, a three-dimensional numerical simulation of turbulent flow is presented. The Reynolds averaged Navier–Stokes (RANS) equations of the fluid flow have been solved numerically using the finite volume method implemented in the FLUENT computer code (see [26] and [33–35]). The SIMPLE (Semi-Implicit Method for Pressure-Linked Equations) procedure was applied for the pressure-velocity coupling. For convection terms, a second-order upwind scheme has been used to interpolate the face values of the different quantities from the cell-averaged values; for the viscous terms, a second-order central scheme was considered. It should be noted that the face values of pressure terms have been evaluated using the PRESTO (PRESsure STaggering Option) method. The boundary conditions and parameters applied for the numerical solution of the problem are summarized in Table 2.

In the next section, we present (1) the results of the configuration C (single swirling jet), (2) the salient results of all configurations (A, B, and C), and (3) those of configuration A. For configuration C, we are interested in the effects of the inclination angle of the diffuser vanes (swirl number) on dimensionless axial temperature and radial velocity, which are experimentally obtained. For configurations A, B, and C, we focus on the effect of balance and imbalance of temperature on the dimensionless temperature profiles in the radial direction. Finally, we consider configurations A and C for presenting the dimensionless profiles of the radial temperature at several axial locations, while quantifying the errors between experiment and prediction. Whenever it was possible, a comparison between the experimental and numerical investigations was produced.

**RESULTS AND DISCUSSION**

As mentioned earlier, previous studies showed that for a range of experimental conditions, heat transfer enhancement is strongly depending on blade angle and it seems that the Reynolds number has no significant influence, whereas when the initial vane angle of the velocity $\alpha$ increases, the jet is more spreading in the radial direction [15]. In this study, the dimensionless temperature and $r-$ and $x-$coordinates were normalized in the forms $T_r = (T - T_u)/(T_0 - T_u)$, $r/D$, and $x/D$, respectively, where $T$ is the jet temperature, $T_u$ is the ambient temperature, and $T_0$ is the maximum temperature of the air blowing at origin.

Figure 7 shows the profile of the dimensionless temperature in the axial direction at different velocity initial angles for the single swirling jet (configuration C). As can be seen, the axial temperature decreases rapidly when $\alpha$ increases. Hence,

---

**Table 1** Number of cells used

<table>
<thead>
<tr>
<th>Grid 1</th>
<th>Grid 2</th>
<th>Grid 3</th>
<th>Grid 4</th>
<th>Grid 5</th>
</tr>
</thead>
<tbody>
<tr>
<td>973,556</td>
<td>1,797,607</td>
<td>1,801,362</td>
<td>1,801,913</td>
<td>1,867,537</td>
</tr>
</tbody>
</table>

---

Figure 6 Sensitivity of dimensionless radial temperature variation to grid density ($k-\varepsilon$ model).
Table 2: Summary of boundary conditions parameters for the solution strategy

<table>
<thead>
<tr>
<th>Parameters</th>
<th>Configuration A</th>
<th>Configuration B</th>
<th>Configuration C</th>
</tr>
</thead>
<tbody>
<tr>
<td>Turbulence intensity</td>
<td>7%</td>
<td>7%</td>
<td>7%</td>
</tr>
<tr>
<td>Underrelaxation Pressure</td>
<td>Pressure = 0.3, Density = 0.8</td>
<td>Momentum = 0.3, Density = 0.3</td>
<td>Pressure = 0.3, Density = 1</td>
</tr>
<tr>
<td></td>
<td>Energy = 1</td>
<td>Other parameters = 0.5</td>
<td>Other parameters = 0.5</td>
</tr>
<tr>
<td></td>
<td>Other parameters = 0.6</td>
<td>Other parameters = 10⁻³</td>
<td>Other parameters = 0.5</td>
</tr>
<tr>
<td>Convergence criteria</td>
<td>Energy = 10⁻⁶</td>
<td>Energy = 10⁻⁶</td>
<td>Energy = 10⁻⁶</td>
</tr>
<tr>
<td></td>
<td>Other parameters = 10⁻³</td>
<td>Other parameters = 10⁻³</td>
<td>Other parameters = 10⁻³</td>
</tr>
<tr>
<td>Number of cells</td>
<td>1,839,831</td>
<td>2,244,126</td>
<td>973,556</td>
</tr>
<tr>
<td>Inlet</td>
<td>U = 6 m.s⁻¹, V = 0, W = 0, T = 90° C ( = 363 K)</td>
<td>Presto method</td>
<td>Pressure inlet</td>
</tr>
<tr>
<td>Pressure walls</td>
<td>No-slip conditions</td>
<td>No-slip conditions</td>
<td>Pressure outlet</td>
</tr>
<tr>
<td>Free surface</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Outlet</td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

the ambient temperature is quickly reached and the area under influence is more important.

Figure 8 presents the comparison of our experimental and predicted results of the radial dimensionless temperature obtained via the standard $k - \varepsilon$ model for different swirl vane angles $\alpha$ for the single swirling jet (configuration C). It can be seen that, only with $\alpha = 60^\circ$, the temperature profile decreases less rapidly, giving an important spreading compared to cases corresponding to $\alpha = 0^\circ$ and $\alpha = 30^\circ$. Also, we note that the vane angle $\alpha = 60^\circ$ improves significantly the thermal homogenization of the flow. To this end, we chose this angle value for all configurations A, B, and C. It is seen from Figure 8 that the standard $k - \varepsilon$ model predicts the shape of the evolution temperature profiles in good agreement with the experimental results for cases tested. However, the standard $k - \varepsilon$ model overestimates the values of the dimensionless mean radial temperature in the region of $0.5 < r/D < 1.5$ and underestimates these values in the region of $1.5 < r/D < 2.3$ for the case $\alpha = 60^\circ$, which confirms the results of Wang and Mujumdar [19]. Indeed, in the swirling flows, due to anisotropy of strain and Reynolds stresses tensors, the standard $k - \varepsilon$ model fails to capture the turbulence effect well. The decrease in the swirl number leads to an improvement in the quality of prediction for both cases, $\alpha = 30^\circ$ and $\alpha = 0^\circ$, respectively [10].

From Figures 7 and 8, we can conclude that when we increase the angle of the blades ($\alpha$) up to $60^\circ$, we find that the swirl number increases. Thus, the axial temperature sharply decreases until it reaches the equilibrium temperature. In addition, the radial temperature decreases slowly and promotes a large spreading of the blown jet for good thermal homogenization.
These results may help to better design diffusers having vanes inclined at an angle of about 60°. These can be used for ventilation applications and air conditioning to reduce harmful effects of axial blowing on occupants and stored products.

In Figure 9, the comparison of the standard $k-\varepsilon$ model results for configuration C with the experimental data is presented. The dimensionless radial main velocity profiles at $\alpha = 60^\circ$ and $\alpha = 30^\circ$ vane inclination and at the two locations $x = 1D$ (left side) and $x = 2D$ (right side) are depicted. We note that for the angle of inclination $\alpha = 30^\circ$, there is an absence of the central circulation zone and a rapid decrease in velocity compared to the case of the angle of inclination $\alpha = 60^\circ$. It should be noted that the decrease of the radial velocity implies the existence of a recirculation zone near the orifice ($r$ close to zero). Moreover, the existence of this zone ($\alpha = 60^\circ$) shifts the maximum of the radial velocity toward large radii. Also, examination of Figure 9 shows that when the inclination angle ($\alpha$) increases up to $60^\circ$, the swirl number increases. Therefore, the strong swirling number induces a central recirculation zone that can delay jet flow. This suggests a large development of the blown jet, which leads to a good thermal homogenization. Moreover, we note that the model $k-\varepsilon$ does not reproduce correctly the experimental results. This result was expected, given the nature of the model.

The comparison of measured data with numerical predictions of the dimensionless radial main velocity profiles at $\alpha = 60^\circ$ vane inclination for two locations $x = 1D$ (left side) and $x = 2D$ (right side) is highlighted in Figure 10. As solution tools, standard $k-\varepsilon$ and Reynolds stress models for turbulence flow were used for this analysis. We note that the $k-\varepsilon$ model underestimates the magnitude of the velocity at the centerline. The Reynolds stress model greatly improves the prediction of this quantity. Note that both models underestimate the minimum of such quantity and provide a good prediction of its maximum value. As can be seen, the predictions of radial dimensionless velocity profiles using the model Reynolds stress model are in generally good agreement with the experimental data.

Both models underestimate the amplitude of the velocity at the centerline because of the inner recirculation zone, as seen in Figure 11. Outside the inner recirculation zone, the standard $k-\varepsilon$ and Reynolds stress models give a better overall agreement with the experimental values. Regarding the maximum predictions, it is clear that the numerical results are much better with the Reynolds stress models.

The radial temperatures profiles associated with configurations A, B, and C at location $x/D = 8$ are presented in Figure 12.
Figure 13  Variation of the dimensionless temperature $T_r$ vs. $x/D$ at the centerline ($r = 0$) for configurations A, B, and C.

One can stress that the temperature varies slowly in the two cases of configurations A and B compared to the single jet (configuration C). Also, the triple swirling jets with configuration A allow a more thermal homogenization with a more important spreading of the resulting jet compared to the B and C configurations (see Figure 12, configuration A). Figure 13 presents and compares the evolution of the dimensionless temperature along the axis of the central line for the A, B, and C configurations. As can be seen, the temperature approaches the ambient temperature ($T_a$) beyond the station $x/D = 8$. The decay is much more visible in the case of configuration A. In this case, the stabilization occurs after the station $x/D = 8$. In the zone $3 < x/D < 8$, configuration B has a relatively stable temperature compared to the configuration of the single swirling jet (see configuration C). Nevertheless, the general trend is quite well matched. For instance, the simulation using the standard $k - \varepsilon$ model underestimates strongly the curves for configurations B and C and overpredicts the curve of configuration A.

The radial distribution of dimensionless temperature at distance from the inlets $x/D = 1$, 3, 5 and 8, respectively, is presented in Figure 14. The standard $k - \varepsilon$ and Reynolds stress models were used to predict the turbulence flow. Along the radial coordinate, the dimensionless temperature profile moves from high values, decreases, and then finally approaches its asymptotic value, which refers to ambient temperature. Some differences from the experimental data were observed in the zone close to the axis for $x/D > 3$. For stations $x/D = 3$, 5, and 8, both models underestimate the maximum value at the centerline. As can be seen, the temperature predictions obtained by Reynolds stress model are generally in good agreement with our experimental data.

The radial distribution of the dimensionless temperature at distance from inlets $x/D = 1$, 3, 5, and 8, respectively for configuration A is shown in Figure 15, which is supplemented by a figure showing the quantification of percentage errors. In this addendum, $E-RSM$ and $E-k-\varepsilon$ denote the percentage of errors with the models RSM and $k - \varepsilon$, respectively. Recall that configuration A consists of a central hot flow blown in the opposite sense of rotation of adjacent fluxes that are less heated than the central flux. Among all the studied configurations, configuration A ensures maximum temperature stability in the radial direction with a fast homogenization compared with other
configurations. The stability starts at station $x/D = 5$ with a 0.2 amplitude for this configuration as shown in Figure 15, and at station $x/D = 8$ with a 0.5 magnitude for configuration C as shown in Figure 13. Returning to Figure 15, we note that along the radial coordinate, the dimensionless temperature profiles have two peaks characterizing the mixing zone. Then they gradually decrease to the asymptotic value that is the ambient temperature. Overall, the predicted results are in good agreement with experimental data but there is a small discrepancy in prediction of the radial development of stability zone. Outside this zone, we note that Reynolds stress model significantly improves the results. Moreover, the quantification of errors shows...
that the RSM model clearly minimizes errors.

In Figures 16 and 17, the radial profiles of dimensionless kinetic turbulence energy \( k^* = k/U_0^2 \) and dissipation rate energy \( \varepsilon^* = \varepsilon/D/U_0^3 \) are plotted at different axial locations. These profiles are presented here to provide a qualitative idea of the turbulence generation processes in the shear layers of the jets. In the case of a single swirling jet (configuration C), we observe one peak near the orifice, and then two peaks far from the orifice corresponding to the location of the shear layer, thereby confirming that homogenization has not yet occurred. For the configuration of triple swirling jets (A and B), the dimensionless profiles of turbulence kinetic energy and its dissipation rate exhibit two distinct peaks at small axial distance \( x/D \). We conjecture that these two peaks indicate approximately the locations of the shear layer for the central jet and peripheral jet, respectively. After mixing of the peripheral and central jets, a single peak remains. This peak gradually moves from the shear layer region toward the axis of the fully developed jet, thus leading to thermal homogenization. Due to its asymmetry in terms of temperature, configuration A has high turbulence intensity at a given station. This finding confirms the fact that in regions near the diffusers, the turbulent kinetic energy is higher for the multiple jets than for the swirling single jet. This is conspicuous in Figures 16 and 17, where \( k^* \) and \( \varepsilon^* \) decrease sharply in cases A and B, compared to the single swirling jet case. It is interesting to note that the simulation achieves easily some quantities such as the energy-dissipation rate, which is notoriously difficult to measure experimentally in laboratory jet flows at small scales.

![Figure 16](image_url) Radial distribution of the dimensionless turbulent energy \( k^* = k/U_0^2 \) at locations \( x/D = 5, 6, 7, \) and 8.

![Figure 17](image_url) Radial distribution of the dimensionless dissipation rate \( \varepsilon^* = \varepsilon/D/U_0^3 \) at locations \( x/D = 5, 6, 7, \) and 8.
Figure 18  Comparison of temperature contours for configurations A, B, and C at the plane $y = 0$.

compared to that of the mean motion. Indeed, special attention should be paid to the effects of spatial resolution on the estimation of such a quantity. For further details, the interested reader can turn to the abundant literature about this topic.

Figure 18 shows contour plots of the temperature for configurations A, B, and C. Note that this figure is composed of two parts: The top part is a three-dimensional illustration of jets, while the bottom part is the temperature mapping in the plane $(x, z)$ at the plane $y = 0$. Through this mapping, we see that thermal homogenization occurs rather sharply in the case of configuration A because of rapid thermal stabilization.

CONCLUSIONS

In this work, an experimental and numerical study of different configurations of blowing multiple swirling jets for use in ventilation applications has been fulfilled. The numerical simulation of the flow and temperature fields has been carried out using the standard $k-\varepsilon$ and the Reynolds stress turbulence models. Based on the investigation conducted for different configurations and parameters, the following conclusions can be made. We have highlighted more improvement of the thermal homogenization of the treated area using multiple swirling jets with an appropriate choice of the position for blowing air. The analysis of the flow features clearly demonstrates that the interaction between swirling jets induces the redistribution of temperature in the mixing zone, while allowing the spreading of the resulting jet. It appears that the central jet plays an important role in the enhancement of the thermal homogenization. From the thermal homogenization viewpoint, with the parametric study of the diffuser geometry, the swirlier vane angle, the number of blowing jets, and the direction of rotation, balance and imbalance in temperature between the central and peripheral jets are adequate means to enhance the quality of thermal homogenization. Thus, when the vanes inclination increases, the axial velocity decreases and the jet spreads radially. Under these conditions, along the flow, the axial temperature decreases and the radial temperature increases. When comparing the evolution of the axial and radial temperature, configuration A shows a better radial stability with an axial decay importance. Of all the configurations studied, configuration A insures maximum and faster radial temperature stability. Overall, obtained quantities with the Reynolds stress model are in better agreement with the experimental data compared to those of the standard $k-\varepsilon$ model.

ACKNOWLEDGMENTS

Fruitful discussions with Gary Manner (University of Artois, France) are gratefully acknowledged. We are also indebted to the editor in chief and anonymous referees for their insightful comments and suggestions that improved the quality of the present work.

NOMENCLATURE

$C_p$  heat capacity, J/kg-K
$C_{u_1}, C_{u_2}$  turbulence model constants
$d$  vane support diameter, m
$D$  inner diameter of one diffuser, m
$e$  spacing between diffusers, m
$E - k - \varepsilon$  errors’ percentage with the $k - \varepsilon$ model
$E - RSM$  errors’ percentage with the RSM model
$G_\theta$  axial flux of tangential momentum
$G_c$ axial momentum flux

$l$ initial turbulence intensity ($= 100(\overline{u^2} + \overline{w^2} + \overline{v^2})/3l^{1/2}/U$) defined in Eq. (14)

$k$ turbulent kinetic energy, $m^2/s^2$

$k^*$ dimensionless turbulent kinetic energy ($= k/U_0^2$)

$l$ turbulence length scale, $m$

$m$ mass flow rate, kg/s

$P$ pressure, Pa

$P_k$ rate of production of turbulent kinetic energy, kg/ms$^3$

$Re_0$ Reynolds number at air blowing origin, dimensionless

$Re_D$ the Reynolds number based on the inner diameter $D$, dimensionless

$r$ radial coordinate of air flow, $m$

$R$ characteristic radius, $m$

$R_b$ radius of the inlet duct, $m$

$R_n$ radius of the centre body, $m$

$S$ swirl number, dimensionless

$T$ temperature of jet, $K$

$T_a$ ambient temperature, $K$

$T_0$ maximum temperature of the air blowing at origin, $K$

$T_r$ dimensionless temperature ($= (T - T_a)/(T_0 - T_a)$)

$u'$ fluctuating velocity, $m/s$

$\rho u_i u_j$ Reynolds stresses, kg/ms$^2$

$U_i$ velocity components, $m/s$

$U_0$ maximum value of the velocity of the air blowing at origin, $m/s$

$U$ mean axial velocity based on flow rate, $m/s$

$W$ mean tangential velocity, $m/s$

$x$ axial coordinate of the air flow, $m$

Greek Symbols

$\alpha$ inclination angle of the vanes

$\varepsilon$ turbulence energy dissipation rate, $m^2/s^3$

$\varepsilon^*$ dimensionless dissipation rate ($= \varepsilon D/U_0^3$)

$\lambda$ thermal conductivity, $W/m.K$

$\mu$ dynamic viscosity, kg/ms

$\mu_t$ turbulent viscosity, kg/ms

$\rho$ air density, kg/m$^3$

$\sigma_l$, $\sigma_c$ turbulent Prandtl numbers, dimensionless

Subscripts

$i, j$ component indices

$t$ turbulent

$\theta$ tangential direction

Superscripts

$\cdot$ time average

$^*$ dimensionless quantity

REFERENCES


Ali Khelil received his diploma in mechanical engineering from Chlef University (Algeria). In 2001, he obtained a master in mechanical engineering. In 2006, he joined Lille 1’s University of Sciences and Technologies, Polytech’Lille, to pursue the degree of doctor in mechanical engineering. He obtained his degree of doctor in 2008. Since 2008, he has been an associate professor at University of Chlef, Algeria. His research activities are on combustion (numerically and empirically), computational fluid dynamics,
and heat transfer. Since 2001, he has participated in several national and international meetings and scientific conferences. He is author and co-author of over 20 referred journal papers.

Hassane Naji is a full professor of mechanical engineering at Artois University (France). He received his undergraduate and graduate education in the Mechanical Engineering Department of Lille 1 University (France). Following research engineer and assistant professor positions at Rouen University and INSA, Rouen, France, he joined Polytech’Lille School of Mechanical Engineering—Lille 1 University as an associate professor until 2010, and then moved to the Artois University (AU). He holds a Ph.D. in physical sciences from the University of Lille 1 (France) and an M.Sc. in thermal-fluid sciences from the National Polytechnic Institute of Lorraine (INPL), Nancy, France. His research and teaching focuses on thermal-fluid sciences and engineering, advanced heat transfer, numerical methods for multiscale simulations (mesoscopic and macroscopic approaches), and applied thermodynamics for engineering. In terms of research, he is the author of more than 100 referred articles in international journals and conferences. Also, he is a referee for many scientific journals.

Mohamed Braikia obtained a master in mechanical engineering from Oran University (Algeria). Currently, he is an associate professor at the University of Chlef, Algeria, since 2002, and he is involved in several projects regarding turbulence and heat transfer. His research works and teaching concern experimental techniques, numerical and experimental thermofluid, computational fluid dynamics, and thermal-fluid sciences. Since 2001, he has participated in several national and international meetings and scientific conferences.

Larbi Loukarfi is a full professor at Chlef University (Algeria) since 2004. He has been Vice-President of Research at Chlef University, and he is involved in several projects regarding turbulence, heat and mass transfer, and renewable energy. Since 1998, he has participated in several national and international meetings and scientific conferences. His research studies and teaching subjects include thermal-fluid science and engineering, fundamentals of advanced energy conversion and combustion, advanced fluid mechanics, and turbulence. He is author and co-author of more than 40 referred journal papers.