CFD Investigation of Thermal Characteristics for a Dual Jet with a Parallel Co-flow

Tanmoy Mondal, Nidhal Hnaien,* Mariem Ajmi, Kouather Ghachem, and Lioua Kolsi

ABSTRACT: A combined turbulent wall jet and offset jet (also known as the dual jet) with and without the presence of a parallel co-flow stream is studied. The standard $k-\omega$ turbulence model is used to predict the turbulent flow. The study focuses on the effects of the co-flow velocity (CFV) on the heat-transfer characteristics of the dual jet flow with the bottom wall maintained at a constant wall temperature. The CFV is varied up to 40% of the jet inlet velocity, and the height of the offset jet is varied from 5 to 11 times the jet width with the inlet Reynolds number taken as 15,000. The heat-transfer results reveal that the local Nusselt number ($N_u$) along the bottom wall exhibits a peak at the immediate downstream of the nozzle exit, followed by a continuous decay in the rest of the converging region before showing a small rise for a short streamwise distance in the merging region. Further downstream, in the combined region, $N_u$ gradually decreases with the downstream distance. Except the merging region, no influence of co-flow is observed in the other two flow zones (converging and combined regions). In the merging region, for a given offset ratio (OR), $N_u$ remains nearly constant for a certain axial distance, and it decreases as the CFV increases. As a result of the increase in the CFV, the average Nusselt number decreases, indicating a reduction in overall convective heat transfer for higher values of the CFV. A regression analysis among the average Nusselt number ($\overline{N_u}$), CFV, and OR results in a correlation function in the form of $\overline{N_u} = e^{7.5383} \times OR^{-0.0073} \times (CFV + 10)^{-0.0204}$ within the range $OR = 5−11$ and $CFV = 0−40\%$.

1. INTRODUCTION

Turbulent jets accompanied by a parallel co-flow are often seen in many practical applications, for example, disposal of waste water in the rivers and oceans, wakes of aircraft, and emission of pollutants in chimneys. The present study focuses on a dual jet flow in the presence of a stream of parallel co-flow. The dual jet flow comprises the wall jet and the offset jet. A wall jet is formed when the fluid issuing from the nozzle flow tangentially along the wall, whereas an offset jet occurs if the nozzle of the jet is at a distance perpendicular to the bottom wall. The wall jet is a special case of the offset jet where the offset height of the nozzle is zero.

Several studies are also found on the flow configuration of a plane wall jet with the presence of a parallel co-flow. A plane wall jet with a stream of co-flow was first studied by Kruka and Eskinazi. Using the hot-wire anemometer, they varied the ratio of co-flow velocity (CFV) to the jet velocity in the range $CFV = 5.5\%$ to $48.5\%$ for a Reynolds number fixed at $Re = 26,270$. Here, CFV is defined as the percentage of the ratio of the CFV to the jet velocity, that is, $CFV = \frac{U_{co}}{U_{jet}} \times 100$. They divided the jet flow into two regions separated at locations of maximum streamwise velocity $U_{\text{max}}$ for the measurement of mean flow and at locations $U'V' = 0$ (where $U'$ and $V'$ are the fluctuating velocity components of the mean velocity) for the statistical quantities. They found a linear relationship between the shear in the free mixing layer and the maximum excess velocity, which was used to calculate the turbulent shear stress from the mean flow quantities. They also found that the frictional velocity at the wall is proportional to $U_{\text{max}}$.

In an experimental study, Irwin studied a co-flowing plane wall jet with the presence of an adverse pressure gradient by varying the CFV up to $38\%$ for $Re = 28,000$. The production of turbulent kinetic energy was found to remain always positive in the outer region of the wall boundary layer. However, the point of zero wall shear stress was found closer to the wall than the maximum velocity. Furthermore, mean velocity profiles in the region close to the wall displayed a logarithmic profile with constants which are similar to those found in the case of the

Received: January 29, 2022
Accepted: July 18, 2022
Published: August 1, 2022
conventional boundary layer flows. Campbell theoretically investigated a plane turbulent wall jet in a co-flowing stream by using an integral method, including the turbulent shear stress, entrainment, and heat transfer. The conservation equations for the incompressible turbulent flows were solved for the average jet flow and heat-transfer properties. The velocity profiles suggested by Escudier and Nicoll were used to obtain the detailed characteristics of the wall jet with co-flow. The analytical results showed a good agreement with the corresponding experimental data when CFV was varied from 5 to 50%.

An experimental study was carried out by Dakos et al. to investigate the flow and heat-transfer characteristics for a plane as well as a curved wall jet in the presence of a co-flow stream with CFV = 57% and \( Re = 30,000 \). According to their experimental measurement, both the jet flows did not achieve the self-similar property due to the presence of the co-flow. The positions of zero shear stress and the zero streamwise heat flux were located closer to the wall due to wall curvature. Furthermore, the shear stress, turbulence intensity, and the turbulent heat fluxes increased in the outer shear layer of the curve jet owing to the extra strain caused by the curvature of the wall.

Ayech et al. numerically investigated the isothermal and non-isothermal wall jets in the presence of a co-flow stream for the different values of CFV ranging from 0 to 20%. They concluded that the length of the potential core decreased with the increase of the CFV. Although the effect of CFV on flow and heat-transfer parameters was found to be negligible in the potential core region, a strong effect of CFV was visible in the established region of the wall jet flow, especially for the higher values of the CFV. Further, Ayech et al. adopted an improved version of the low Reynolds number model proposed by Herrero et al. for predicting a plane isothermal as well as a plane non-isothermal wall jet with the presence of a directed co-flow stream with CFV varying up to 40%. They concluded that with the increase of the deflection angle of the co-flow, the growth of the jet half-width, the maximum temperature, and the temperature half-width increased.

Compared to the wall jet, studies on the offset jet with the presence of a co-flow are sparse. Hoch and Jiji used a combined theoretical and experimental approach to investigate the effects of the CFV on the flow field of an offset jet for the CFV varied up to 27%. An integral formulation based on the entrainment approximation was used to predict the offset jet characteristics for different free stream velocities. A good agreement between the theoretical predictions and the experimental observations substantiated the hypothesis that the entrainment mechanism in the offset jet flow remained unaffected by the pressure field. The same authors (Hoch and Jiji) studied the thermal characteristics of the offset jet flow both theoretically and experimentally, and the influence of CFV on the decay of maximum streamwise temperature was found to be insignificant.

Although many investigators studied the flow characteristics of the dual jet flow in detail, a limited attention was paid so far on the heat-transfer aspects of the dual jet. Wang and ‘Tan were the first investigators who conducted an experiment on a combined wall and offset jet (also popularly known as a dual jet) with the offset ratio (OR) of 2 times the jet width. Compared to the fluid flow, the heat-transfer analysis for the dual jet is sparse. With the use of the standard \( k–\varepsilon \) turbulence model, Vishnuvardhanarao and Das numerically examined the heat-transfer characteristics of the dual jet flow considering the same OR used by Wang and Tan. They used two boundary conditions, that is, constant heat flux and constant wall temperature. They varied \( Re \) in the range 10,000–40,000 and velocities of both the wall and offset jets from 0.25 to 1.0 with an interval of 0.25. A few years later, with the use of standard \( k–\varepsilon \), Kumar numerically solved a problem on the dual jet with the bottom wall kept at isothermal as well as at constant heat flux boundary conditions. They varied the OR in the range OR = 3–15 for a constant Reynolds number \( Re = 15,000 \). As an important result, two peaks were found in the profiles of local heat flux as well as in the profiles of local Nusselt number variation when the OR was greater than 5 (i.e., \( OR > 5 \)).

Later, Hnaien et al. investigated the thermal characteristics of a dual jet with an oblique wall. They noticed that an increase in oblique angle resulted in a decrease of Nusselt number. In a contemporary work, Hnaien et al. studied the effects of the OR on the thermal characteristics of a dual jet with uniform heat flux boundary condition for a plane wall surface in the range \( OR = 5–20 \) and \( Re = 10,000–40,000 \). They concluded that the convective heat transfer was intensified when the OR decreased and the Reynolds number increased. Later on, Assoudi et al. studied the heat-transfer characteristics of the dual jet using the Reynolds stress and the standard \( k–\varepsilon \) turbulence models. When comparing the results with the experimental data of Wang and Tan, the Reynolds stress model performed better than the standard \( k–\varepsilon \) model. Moreover, the rate of decay of temperature was found to increase with the decrease in the velocity ratio.

Recently, Singh et al. studied the hydrodynamics and heat transfer for the dual jet flowing over a wavy wall by varying the amplitude and number of cycles of the sinusoidal wave equations. The heat-transfer rate gradually increased up to a certain amplitude ratio and then decreased with a maximum increase of 12% heat-transfer rate for an amplitude of 0.5. With the use of wavy wall, the overall heat-transfer enhancement was found to achieve 23.27% over the corresponding wall jet without wavy wall.

According to the above literature survey, although studies are available on flow configurations of both the wall jet and the offset jet with the presence of a co-flow, no research was conducted on the flow configuration of the dual jet accompanied by the co-flow. The presence of an external stream prohibits the surrounding fluid to enter into the core region of the jet and thereby the growth of the jet is hindered. This may in turn affect the convection heat transfer from the heated wall to the surrounding fluid significantly; however, none of the research papers in the past brought this research gap into attention. The present study thus considers a dual jet flow in the presence of a parallel co-flow with a specific focus given on the influence of the cold co-flow stream on the convective heat-transfer field for the dual jet with a heated bottom wall. To capture the influence of the co-flow on the heat-transfer characteristics to the dual jet comprehensively, the CFV and the OR are varied over a wide range of CFV and OR with 10% ≤ CFV ≤ 40% and 5 ≤ OR ≤ 11 for a fixed value of Reynolds number, that is, \( Re = 15,000 \). It is worthy to mention here the experimental study of Hoch and Jiji on a single offset jet with CFV varying up to 27%, which is close to the maximum CFV value considered in the current study. Moreover, both Holland and Liburdy and Kim et al. conducted their experiment on a single offset jet without the presence of a co-flow by varying the OR within the range OR = 2.5–10.5. In a computational study on heat transfer to a dual jet, Kumar varied the OR from 3 to 15.
2. PROBLEM DESCRIPTION

Figure 1 represents a schematic diagram for a combined two-dimensional flow configuration of a dual jet with a parallel co-flow just above the offset jet. Here, the lower jet is the wall jet and the upper one is the offset jet. Two turbulent cold jets with the same inlet velocity \( u_0 \) and width \( w \) are used to cool down the heated bottom wall maintained at a constant wall temperature. The height of the lower extremity of the offset jet nozzle from the horizontal wall, that is, \( Y = 0 \), is denoted as or (as shown in Figure 2). At the inlet of the flow domain, the velocity of the co-

flow \( u_{CF} \) is kept lower than the jet inlet velocity and the temperature is the same as the jet inlet temperature. The nozzle exit plane is represented by the \( y \) axis, that is, \( X = 0 \).

Since the entrainment of the wall jet is blocked by the offset jet, the two jets begin to attract each other after issuing from the nozzle, and eventually, they merge each other at the merge point (MP). This phenomenon is popularly called as the Coanda effect.\(^{22}\) Further downstream of MP, the interaction of the two merged jets persists until the combined point (CP), where the flow is seen to be analogous to the wall jet. Several flow zones are formed in the dual jet flow: the converging zone (between MP and the nozzle exit plane, i.e., \( X = 0 \)), the merging zone (between MP and CP), and the combined zone (downstream of CP or rest of the flow domain).

3. MATHEMATICAL FORMULATION

3.1. Governing Equations. Turbulent flow under the present study is considered as two-dimensional (2D) and steady; the fluid is assumed as incompressible without the presence of body forces. The RANS (Reynolds-averaged Navier–Stokes equations) is used for numerical modeling. According to Figure 2, the length of the nozzle (\( l = 150 \) mm) is considered to be much larger than its width (\( w = 12.5 \) mm). Therefore, the influence of spanwise direction (\( Z \) direction) can be ignored to ensure the flow to be 2D. For the turbulence closure, the two-equation standard \( k-\omega \) turbulence model is utilized. The dimensional form of the governing equations for the \( k-\omega \)-based RANS can be written in Cartesian tensor representation as follows

\[
\frac{\partial}{\partial x_i}(\rho u_i) = 0 \tag{1}
\]

\[
\frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \right) \right] + \frac{\partial}{\partial x_j} (-\rho u_i u_k') \tag{2}
\]

The energy equation is given by the following equation

\[
\frac{\partial}{\partial x_i} \left[ \rho (\rho E + p) \right] = \frac{\partial}{\partial x_j} \left( \lambda_{\text{eff}} \frac{\partial T}{\partial x_j} + u_i (\tau_{ij})_{\text{eff}} \right) + S_h \tag{3}
\]

The \( E \) term is the total energy, \( \lambda_{\text{eff}} \) is the effective thermal conductivity, and \((\tau_{ij})_{\text{eff}}\) is the deviatoric stress tensor defined by

\[
(\tau_{ij})_{\text{eff}} = \mu_{\text{eff}} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \frac{\partial u_k}{\partial x_k} \right) \tag{4}
\]

\[
\lambda_{\text{eff}} = \lambda + \frac{C_p \mu_t}{Pr_t} \tag{5}
\]

where \( \lambda \) and \( Pr_t \) (constant value 0.85) are the thermal conductivity and turbulent Prandtl number, respectively. The Boussinesq hypothesis which essentially relates the Reynolds stress terms present in eq 2 with the mean velocity gradients is utilized for the turbulence closure as presented in eq 6.
Figure 3. Grid size details and boundary conditions.

\[-\rho u_i u_j = \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \rho k \right) \delta_{ij} \]  \hspace{1cm} (6)

The turbulent kinetic energy \( k \) and the rate of specific dissipation of turbulent kinetic energy \( \omega \) are calculated from eqs 7 and 8, respectively.

\[ \frac{\partial}{\partial x_i} \left( \rho u_i u_j \right) = \frac{\partial}{\partial x_j} \left( \Gamma_k \frac{\partial k}{\partial x_j} \right) + G_k - Y_k + S_k \]  \hspace{1cm} (7)

\[ \frac{\partial}{\partial x_i} \left( \rho \omega u_j \right) = \frac{\partial}{\partial x_j} \left( \Gamma_\omega \frac{\partial \omega}{\partial x_j} \right) + G_\omega - Y_\omega + S_\omega \]  \hspace{1cm} (8)

In eqs 7 and 8, the terms \( G_k \) and \( G_\omega \) denote the generation of \( k \) and \( \omega \), respectively. \( \Gamma_k \) and \( \Gamma_\omega \) correspond the effective diffusivity for \( k \) and \( \omega \), respectively. \( Y_k \) and \( Y_\omega \) represent the dissipation of \( k \) and \( \omega \), respectively; \( S_k \) and \( S_\omega \) are the source terms. \( \Gamma_k \) and \( \Gamma_\omega \) are given by the following expressions:

\[ \Gamma_k = \mu + \frac{\mu_t}{\sigma_k} \]  \hspace{1cm} (9)

\[ \Gamma_\omega = \mu + \frac{\mu_t}{\sigma_\omega} \]  \hspace{1cm} (10)

\[ \mu_t = \alpha^* \frac{\rho k}{\omega} \]  \hspace{1cm} (11)

Here, \( \sigma_k = \sigma_\omega = 2 \) denote the turbulent Prandtl numbers, respectively, for \( k \) and \( \omega \). \( \alpha^* \) is calculated as follows:

\[ \alpha^* = \alpha^{*\infty} \left( 1 + \frac{Re_t}{Re^{*\infty}} \right) \]  \hspace{1cm} (12)

where, \( Re_t = \frac{\rho k}{\mu \omega} \) \hspace{1cm} (13)

\[ \alpha^{*\infty} = \frac{\beta}{3} \]  \hspace{1cm} (14)

Here, \( R_k = 6 \) and \( \beta = 0.072 \) are constants, and \( \alpha^* = \alpha^{*\infty} = 1 \) for \( k-\omega \) model with high Reynolds number. The terms \( G_k \) and \( G_\omega \) respectively, in eqs 7 and 8 can be obtained by:

\[ G_k = -\rho u_i u_j \frac{\partial u_j}{\partial x_i} \]  \hspace{1cm} (15)

\[ G_k = \mu_t S^2 \]  \hspace{1cm} (16)

\[ S = \sqrt{2 \over \sigma_\omega \sigma_k} \]  \hspace{1cm} (17)

\[ G_\omega = \alpha^* \frac{\omega}{k} G_k \]  \hspace{1cm} (18)

\[ \alpha = \frac{\alpha^{*\infty}}{\alpha^*} \left( \frac{\alpha_0 + Re_t/Re^{*\infty}}{1 + Re_t/Re^{*\infty}} \right) \]  \hspace{1cm} (19)

Here, \( R_\omega = 2.95, \alpha_\omega = 0.52, \) and \( \alpha_0 = 1/9 \) are constants. \( Re_t \) and \( \alpha^* \) are, respectively, given by eqs 13 and 12 with \( \alpha = \alpha^* = 1 \) for \( k-\omega \) model with high Reynolds number.

\textbf{3.2. Geometric Configuration.} Figure 2 shows the geometric configuration of the combined wall and offset jet flow adopted for the present computational study. The nozzle’s width and length are taken as \( w = 12.5 \) mm and \( l = 150 \) mm, respectively. The co-flow is ejected parallel to the direction of the main jets and is located just above the offset jet nozzle. The height of the co-flow stream from the upper extremity of the offset jet nozzle is kept always fixed at 20 times the jet width (i.e., \( 20 \times w \)), whereas the OR and the CFV are varied in the range \( 5 \leq OR \leq 11 \) and \( CFV = 0-40\% \), respectively. Here, the OR is defined as the ratio of the height of the lower extremity of the offset jet nozzle to the jet width, that is, \( OR = or/w \), and the CFV is defined as \( CFV = \frac{\text{CFV}}{100} \times 100 \). The total dimension of the computational domain is taken as \( 70W \times 40W \) (along the X and Y axes, respectively), ensuring a smooth development of the flow throughout the domain.

First of all, the present numerical model is validated on the geometric configuration of Pelfrey and Liburdy\(^{15}\) for single offset jet (zero wall jet velocity) without the co-flow (CFV = 0%) and for an OR = 6.5. For the rest of the calculations, the geometric configuration of the combined wall and offset jet flow...
is considered to study the effect of co-flow on the mean flow behaviors.

3.3. Grid Generation and Boundary Conditions. As clearly shown in Figure 3, a uniform mesh is adopted along the offset jet, the wall jet, the co-flow, and the nozzle spacing with mesh spacing \( a = 0.08 \). For the lateral side, along the vertical direction, the mesh is non-uniform with spacing \( a = 0.3 \) and an expansion ratio \( e = 1.05 \). For the horizontal wall, the mesh spacing and the expansion ratio are 0.3 and 1.01, respectively. Note that all the opposite segments have the same grid size details.

The uniform velocity profile is adopted along the wall jet, the offset jet, as well as the co-flow (Velocity Inlet boundary condition). Along the horizontal plate and the nozzle spacing,
the Wall (no slip condition) boundary condition is adopted. At the lateral side and the flow outlet, we adopted, respectively, Pressure Inlet and Pressure Outlet boundaries conditions. Note that for the validation case (single offset jet without co-flow), no modification is performed on the mesh size; the only modification concerns the boundary condition at the wall jet as well as on the co-flow (the boundary condition changed from Velocity Inlet to Wall). The boundary and inlet conditions are presented in Table 1.

For the thermal boundary conditions, the bottom wall is maintained at a constant wall temperature \(T_{w} = 310\) K. On the other hand, the temperature at the jet inlet as well as at the inlet of the co-flow is isothermal and equals to the ambient temperature, that is, 300 K. At the entrainment side or on the top boundary of the computational domain, the temperature equals to the ambient temperature. At the exit boundary, the temperature gradient normal to the plane is zero. Therefore, the temperature at the exit plane is extrapolated from the solution inside the computational domain.

3.4. Numerical Schemes and Methodologies. To solve the governing equations (eqs 1–5), the CFD (computational fluid dynamics) software “ANSYS FLUENT” is utilized. The diffusive terms of the governing equations have been discretized by using the central difference scheme, and the second-order UPWIND scheme has been utilized to discretize the convective terms of the governing equations. To couple the pressure and velocity, the SIMPLEC algorithm is used. The disposal method of Gauss–Siedel associated with a relaxation technique is used to solve the resulting tri-diagonal matrix. The convergence criterion is set as \(10^{-5}\).

3.5. Grid Independence Test and Validation. In Figure 4, the \(U\)-velocity profile at various axial locations, that is, \(X = 3, 6, 9,\) and 12, in Figure 4a–d, respectively, are compared with the experimental results of Pelfrey and Liburdy\(^{23}\) for an offset jet with \(OR = 6.5\). In the same plots, three different mesh sizes with cell counts 42,000, 90,171, and 159,600 are considered to perform the grid independence test. As noticed in Figure 4, the \(U\) velocity profiles corresponding to mesh sizes 90,171 and 159,600 almost overlap with other, and both the curves show a close match with the experimental data of Pelfrey and Liburdy. However, a significant difference is found when the mesh size is increased from 42,000 to 90,171. Therefore, the mesh size with 90,171 quadratic cells is considered for the rest of the computations.

To justify the selection of the standard \(k–\omega\) for the present numerical simulation, the \(U\) velocity profile shown in Figure 4a is predicted for six different turbulence models (standard \(k–\varepsilon\), RNG \(k–\varepsilon\), realizable \(k–\varepsilon\), standard \(k–\omega\), SST \(k–\omega\), and Reynolds stress model) in Figure 5 for mesh size 90,171 along with the corresponding experimental data of Pelfrey and Liburdy.\(^{23}\) It can be noticed in Figure 5 that the numerical predictions of the current study match fairly well with the experimental data of Pelfrey and Liburdy for the \(k–\omega\) standard model, while some difference is observed for the other model. Therefore, for the rest of the computations, the standard \(k–\omega\) model is used.

The validation of the heat-transfer results is presented in Figure 6, depicting the comparison of local Nusselt number \(Nu\) with Song et al. 2000 results for an offset jet with \(OR = 2.5\) and 5. Reprinted (Adapted or Reprinted in part) with permission from [Song, H. B.; Yoon, S. H.; Lee, D. H. Flow and heat transfer characteristics of a two-dimensional oblique wall jet. *Int. J. Heat Mass Transfer* 2000, 43, 2395–2404.]. Copyright [2000/Lioua Kolsi] [International Journal of Heat and Mass Transfer/Lioua Kolsi].

4. RESULTS AND DISCUSSION

4.1. Streamline Patterns. To demonstrate the flow pattern in the region closer to the exit of the two jets, the streamlines are plotted. Flow patterns in the recirculation region are demonstrated through streamlines in Figure 7a–c for CFV = 0, 20, and 40%, respectively, and a fixed \(OR = 7\) as expected, all recirculation zones in Figure 7 contain two primary vortices positioned one above the other: the clockwise rotating upper vortex is formed on the offset jet side, while the counter-clockwise rotating lower vortex is formed on the wall jet side.
The locations of upper ($X_{uvc}$, $Y_{uvc}$) and lower ($X_{lvc}$, $Y_{lvc}$) vortex centers are indicated in each plot of Figure 7. At the vortex center, both streamwise and transverse velocities are zero. For the dual jet with OR = 7, the upper vortex centers are found at ($X_{uvc}$, $Y_{uvc}$): (3.60, 5.03), (3.86, 5.18), and (4.01, 5.34) and the lower vortex centers are found at ($X_{lvc}$, $Y_{lvc}$): (3, 2.08), (3.28, 2.12), and (3.58, 2.17) for CFV = 0, 20, and 40%, respectively. These results suggest that the axial as well as the transverse locations of both vortices in the recirculation zone moves to higher locations as the magnitude of the CFV is increased.

The effects of the co-flow are markedly visible in Figure 7. The direction of streamlines above the offset jet region for CFV = 0% in Figure 7a is almost vertically downward due to the entrainment of surrounding fluid toward the jets. However, for CFV = 20 and 40% in Figure 7b,c, respectively, the streamlines above the offset jet are almost horizontal with a slight deflection toward the bottom wall similar to the streamline patterns for flow past a 2D bluff body. In this case, the nozzle plate between the two jets behaves like a 2D bluff body. Additionally, a close inspection on all the streamline plots in Figure 7 suggests that for CFV = 0% in Figure 7a, the size of the upper vortex is relatively greater than that of the lower vortex due to the unequal entrainment of the surrounding fluid by the two jets—the upper offset jet entrains more fluid than the lower wall jet owing to the presence of the bottom wall. However, with the increase of CFV to CFV = 40% in Figure 7c, the two vortices are found to be almost equal in size, indicating that the size of the lower vortex in the recirculation zone increases as the CFV is increased. The reason is that as the CFV increases, the exchange of momentum between the offset jet and the co-flow stream decreases, and thereby, the rate of decay of offset jet velocity with the
longitudinal distance decreases with increasing CFV. As a result, the wall jet entrains more fluid into the recirculation zone, leading to an increase in the lower vortex size with a decrease in the CFV.

4.2. Mean Velocity Profiles. In order to show the development of various shear layers present in the flow field, the profiles of the streamwise velocities are shown in Figure 8 at six longitudinal locations within the range 0 £ X £ 30 for various values of CFV at a fixed OR = 7.

Two peaks (an upper peak due to the offset jet and a lower peak due to the wall jet) are distinctly visible in the U-velocity profile for each case of CFV up to the longitudinal distance X = 14, indicating the presence of four shear layers: the outer (layer 1) and inner (layer 2) shear layers for the offset jet, the outer shear layer for the wall jet (layer 3), and the wall boundary layer (layer 4), as illustrated schematically in Figure 1.

In the outer edge of layer 1, the U-velocity profile is vertically straight for CFV = 0% but is little bent for other values of CFV in order to achieve the upstream flow conditions. In the neighborhood of the jet exit plane, at X = 3 and 5, the negative U-velocity confirms the existence of a recirculation region between the two jets. At X = 3, all the U-velocity profiles almost coincide with each other up to the transverse distance Y = 8 (i.e., up to layer 2). However, all the U-velocity profiles gradually begin to differ from one another as the downstream distance increases.

From X = 14 onward, the two U-velocity peaks eventually disappear and only a single peak survives in the further downstream at X = 30 for all values of CFV, except CFV = 40%, indicating that the two jets for CFV = 40% are yet to combine together at the combined point (CP). All velocity profiles except CFV = 40% at X = 30 are found to resemble a wall jet-like flow.

As noticed in Figure 8, the magnitude of the U-velocity peak in layer 3 at a given longitudinal location is greater than that in layer 2. This is because of the higher exchange of momentum of the offset jet in the transverse direction. Also, both the peaks gradually decrease as the longitudinal distance increases with the U-velocity peak in layer 2 decreasing faster than that in layer 3, indicating that the combined flow approaches to the flow structure similar to the wall jet as the flow moves in further downstream direction. The transverse position of the U-velocity peaks in layer 2 gradually disappears with axial distance, while in layer 3 remains almost unchanged. This increase in CFV strongly inhibits the offset jet to deflect to the wall jet.

4.3. Lengths of Different Flow Zones and Their Correlation Functions. Figure 9 represents the variation of axial lengths of different flow zones in the flow field of dual jet for CFV = 0–40% and OR = 5–11. The variations of axial length for the converging zone L_{conv}, merging zone L_{merg} and combined zone L_{comb} with respect to the CFV are illustrated in Figure 9a–c, respectively. The lengths L_{conv}, L_{merg} and L_{comb} are calculated based on the axial distances of the merging point (X_{mp}), the combined point (X_{cp}), and the axial length of the computational domain (X = 70) as given in eqs 20–22, respectively.

\[ L_{conv} = X_{mp} \]  
\[ L_{merg} = X_{cp} - X_{mp} \]  
\[ L_{comb} = 70 - X_{cp} \]

The computationally calculated values of L_{conv}, L_{merg} and L_{comb} are presented in the tabular form in Table 2 within the range 5 £ OR £ 11 and 0 £ CFV £ 40. Based on the computational data in Table 2, a regression analysis is carried out to establish a correlation function that explicitly relates each of the lengths with the CFV and the OR for which the current computation is carried out.

Table 2. Comparison between Correlation and Numerical Results (the Correlation Values are between (...) )

| OR | CFV | L_{conv} | L_{merg} | L_{comb} |
|----|-----|----------|----------|----------|
| 5  | 0   | 5.28 (5.07) | 8.1 (8.06) | 56.62 (60.05) |
| 7  | 0   | 7.6 (7.34) | 7.83 (8.06) | 54.57 (56.71) |
| 9  | 0   | 9.69 (9.68) | 7.74 (8.06) | 52.57 (53.37) |
| 11 | 0   | 11.65 (12.07) | 7.77 (8.06) | 50.58 (50.02) |
| 5  | 10  | 5.49 (5.43) | 8.41 (9.09) | 56.1 (57.24) |
| 7  | 10  | 7.88 (7.87) | 8.17 (9.09) | 53.95 (53.90) |
| 9  | 10  | 10.12 (10.38) | 8.16 (9.09) | 51.72 (50.56) |
| 11 | 10  | 12.25 (12.94) | 8.26 (9.09) | 49.49 (47.21) |
| 5  | 20  | 5.67 (5.79) | 9.44 (8.30) | 54.89 (54.43) |
| 7  | 20  | 8.3 (8.39) | 9.34 (9.03) | 52.26 (51.09) |
| 9  | 20  | 10.81 (11.07) | 9.57 (9.81) | 49.62 (47.75) |
| 11 | 20  | 13.26 (13.80) | 10.04 (10.65) | 46.7 (44.40) |
| 5  | 30  | 5.92 (6.15) | 10.97 (10.91) | 53.11 (51.62) |
| 7  | 30  | 8.82 (8.90) | 11.32 (11.85) | 49.86 (48.28) |
| 9  | 30  | 11.73 (11.74) | 12.13 (12.86) | 46.14 (44.94) |
| 11 | 30  | 14.62 (14.64) | 13.42 (13.96) | 41.96 (41.59) |
| 5  | 40  | 6.16 (6.49) | 13.34 (14.31) | 50.5 (48.81) |
| 7  | 40  | 9.46 (9.40) | 14.85 (15.53) | 45.69 (45.47) |
| 9  | 40  | 12.83 (12.39) | 17.45 (16.86) | 39.72 (42.13) |
| 11 | 40  | 16.28 (15.46) | 21.4 (18.30) | 32.32 (38.78) |

average error 2.42% 3.42% 4.83%
As can be observed from eqs 23–26 that $L_{\text{conv}}$ and $L_{\text{merg}}$ are found to be the exponential and polynomial functions of CFV and OR, whereas $L_{\text{comb}}$ is the combined linear function of CFV and OR. Along with the computed values, Table 2 also displays the values of $L_{\text{conv}}$, $L_{\text{merg}}$, and $L_{\text{comb}}$ calculated from the correlation functions. The percentage error of these values with respect to the computationally obtained value is calculated, and the average of the percentage error is found to be 2.42, 3.42, and 4.83% for $L_{\text{conv}}$, $L_{\text{merg}}$, and $L_{\text{comb}}$, respectively. It is evident from Figure 9 that both $L_{\text{conv}}$ and $L_{\text{merg}}$ increase with increasing CFV, whereas $L_{\text{comb}}$ displays a decreasing trend with CFV for a given OR. These results suggest that the converging length as well as the merging length is elongated, while the combined length is shortened as the CFV is increased.

4.4. Distribution of the Local Nusselt Number. The distribution of the local Nusselt number $N_u_x$ for various co-flow velocities CFV at OR = 5, 7, 9, and 11 is shown in Figure 10a–d, respectively. It is found from Figure 10 that all the $N_u_x$ profiles attain to a maximum value immediately after the jet exit and then sharply decrease to a minimum value in the converging region. Thereafter, a small rise in $N_u_x$ is observed for a short longitudinal distance in the merging region before it monotonically decreases with the downstream direction. The increase in $N_u_x$ in the converging region is due to the merging of two cold jets after discharging from the nozzles. The small rise in $N_u_x$ in the merging region can be attributed to the interaction of the two
jets. The decay in \( \text{Nu} \) in the wall jet region is indicative of the development of a thermal boundary layer. It can also be observed from Figure 10 that the all profiles of \( \text{Nu} \) almost overlap with each other in the converging as well as in the combined regions, showcasing that the effects of the co-flow on the convection heat transfer remain insignificant in the converging and in the far downstream of the jet development region. It is interesting to note that for a given OR with the increase in CFV, \( \text{Nu} \) decreases in the merging region, demonstrating a low convective heat-transfer rate for co-flow streams with higher inlet velocity.

4.5. Average Nusselt Number. In order to qualitatively examine the influence of the OR on the rate of convective heat transfer from the heated wall to the flow of two cold jets for various values of co-flow velocities, the average Nusselt number is plotted for in Figure 11, respectively. In the present study, the average Nusselt number is calculated based on the formula

\[
\overline{\text{Nu}} = \frac{1}{L_w} \int_{0}^{L_w} \text{Nu} \, dx,
\]

where \( L_w \) represents the axial length of the computational domain and \( \text{Nu}_l \) is the local Nusselt number.

As can be seen in Figure 11, for a given value of OR, \( \overline{\text{Nu}} \) decreases with an increase in CFV, indicating that the convection heat transfer is greater for lower values of CFV. As the CFV increases, the momentum exchange between the offset jet and the co-flow stream decreases, causing the offset jet to attract the wall jet more strongly. As a result, the interaction between the wall jet with the bottom wall decreases with increasing CFV. Thus, the average Nusselt number decreases as the CFV increases.

4.6. Correlation Functions for the Average Nusselt Number. In order to estimate the value of the average Nusselt number \( \overline{\text{Nu}} \) for a given value of the CFV and the OR for a dual jet flow, a correlation function that can represent \( \overline{\text{Nu}} \) as a function of OR and CFV is derived. For this purpose, a linear regression analysis is performed based on the present computational values of \( \overline{\text{Nu}} \) for different values of CFV and OR as given in Table 3. On the basis of regression analysis, the \( \overline{\text{Nu}} \) is found to vary with CFV and OR following the correlation function:

\[
\overline{\text{Nu}} = e^{7.5383 \times \text{OR}^{-0.00073} \times (\text{CFV} + 10)^{-0.0204}},
\]

which is valid in the range \( \text{CFV} = 0–40\% \) and \( \text{OR} = 5–11 \).

The curves of \( \overline{\text{Nu}} \) versus OR variation for different values of CFV are plotted in Figure 11, showing that for a given value of OR, the values of \( \overline{\text{Nu}} \) decreases with the increase in CFV. This is because the lateral force exerted by the fluid on the bottom wall is higher for lower values of CFV. Table 3 also contains the values of \( \overline{\text{Nu}} \) obtained from the above correlation function for \( \overline{\text{Nu}} \), displaying an excellent concordance between the computation and the correlation function with an average error of 0.42%. The average error is obtained by taking the average of percentage deviation between the correlation data and the numerical data for the average Nusselt number in Table 3.

5. CONCLUSIONS

The present work involves a numerical investigation on a dual jet (a coupled wall jet and offset jet flow) accompanied by a co-flow stream parallel to the jets’ centerlines is solved. Although the flow configurations of a single wall jet and a single offset jet with the presence of a co-flow have been studied separately in the past, no previous research is found for the flow configuration of a dual jet with the presence of a co-flow. The main focus of the current study is to investigate the influence of the co-flow on the convective heat-transfer phenomenon of a dual jet flow exposed in a co-flow surrounding.

A CFD package “ANSYS FLUENT” is used for numerical simulations. A standard \( k-\omega \) model with the Reynolds number \( Re = 15,000 \) is used for predicting the turbulent field. The bottom wall is maintained at a constant wall temperature. The heat-transfer results are obtained by varying both the CFV and the OR within an appreciable range for both the parameters to analyze the convection heat-transfer characteristics for the dual jet flow in conjunction with a co-flow more elaborately and accurately: CFV is varied in the range 0% ≤ CFV ≤ 40%, and OR

| OR | CFV | \( \text{average Nu} \) |
|----|-----|------------------|
| 5  | 0   | 1778 (1790)      |
| 7  | 0   | 1783 (1790)      |
| 9  | 0   | 1785 (1790)      |
| 11 | 0   | 1785 (1789)      |
| 5  | 10  | 1766 (1765)      |
| 7  | 10  | 1771 (1765)      |
| 9  | 10  | 1774 (1764)      |
| 11 | 10  | 1774 (1764)      |
| 5  | 20  | 1755 (1751)      |
| 7  | 20  | 1759 (1750)      |
| 9  | 20  | 1760 (1750)      |
| 11 | 20  | 1760 (1750)      |
| 5  | 30  | 1745 (1740)      |
| 7  | 30  | 1744 (1740)      |
| 9  | 30  | 1742 (1740)      |
| 11 | 30  | 1738 (1739)      |
| 5  | 40  | 1729 (1732)      |
| 7  | 40  | 1724 (1732)      |
| 9  | 40  | 1718 (1732)      |
| 11 | 40  | 1710 (1732)      |

average error 0.42%
is varied from 5 to 11. Based on the computational results of the current study, the following conclusions are attained:

(i) For a given value of OR, the variation of the local Nusselt number $N_u$ for different values of CFV exhibits a similar trend. $N_u$ initially increases to a maximum value in the converging region and then decreases with the axial distance before displaying a small rise for a certain distance in the merging region. In the combined region, the local Nusselt number keeps decaying with the downstream length similar to the phenomenon of development of the thermal boundary layer for a classical wall jet flow.

(ii) The effect of the CFV on $N_u$ is found to be subdued in the converging as well as in the combined zone, but an appreciable change in $N_u$ is observed in the merging zone with a decrease in $N_u$ as the OR increases. The same trend of distribution of the local Nusselt number is observed when the Reynolds number is varied in the range $15,000 \leq Re \leq 45,000$ for a given value of CFV with an increase in $N_u$ as Re increases.

(iii) The average Nusselt number for all the ORs considered is found to decrease with the increase in the CFV, indicating that the convection heat transfer decreases as the CFV increases.

(iv) A regression analysis based on the computational values corresponding to the different values of CFV and OR results in a correlation function in the form $Nu = \exp{2.5383 \times OR^{0.0073}} \times (CFV + 10)^{-0.2044}$ for $0 \leq CFV \leq 40\%$ and $5 \leq OR \leq 11$. This correlation function can be used effectively to calculate any intermediate values of CFV and OR with the range specified without performing the numerical simulations as well as experimental measurements.

#### AUTHOR INFORMATION

**Corresponding Author**

Nidhal Hnaien — Laboratory of Thermal Processes, Research and Technology Centre of Energy, Hammam Lif 5050, Tunisia; orcid.org/0000-0001-9888-389X; Email: nidhalhnaien@gmail.com

**Authors**

Tanmoy Mondal — Department of Applied Mechanics, Motilal Nehru National Institute of Technology Allahabad, Prayagraj, Uttar Pradesh 211004, India

Mariem Ajmi — Energy Engineering Department, Laboratory of Metrology and Energy Systems, University of Monastir, National Engineering School of Monastir, Monastir 5019, Tunisia

Kouther Ghachem — Department of Industrial Engineering and Systems, College of Engineering, Princess Nourah Bint Abdulrahman University, Riyadh 11671, Saudi Arabia

Lioua Kolsi — University of Ha’il, Ha’il City 81451, Saudi Arabia

Complete contact information is available at: https://pubs.acs.org/10.1021/acsomega.2c00609

**Notes**

The authors declare no competing financial interest.

#### ACKNOWLEDGMENTS

Princess Nourah bint Abdulrahman University Researchers Supporting Project number (PNURSP2022R41), Princess Nourah bint Abdulrahman University, Riyadh, Saudi Arabia.

#### REFERENCES

1. Kumar Rathore, S.; Kumar Das, M. Effect of freestream motion on heat transfer characteristics of turbulent offset jet. *J. Therm. Sci. Eng. Appl.* 2016, 8, 011021.

2. Kruka, V.; Eskinazi, S. The wall-jet in a moving stream. *J. Fluid Mech.* 1964, 20, 555–579.

3. Irwin, H. P. A. Measurements in a self-preserving plane wall jet in a positive pressure gradient. *J. Fluid Mech.* 1973, 61, 33–63.

4. Campbell, J. F. Turbulent wall jet in a coflowing stream; No. NASA-TN-D-8025, 1975.

5. Escudier, M. P.; Nicolli, W. B. The entrainment function in turbulent boundary-layer and wall-jet calculations. *J. Fluid Mech.* 1966, 25, 337–366.

6. Dakos, T.; Verriopoulos, C. A.; Gibson, M. M. Turbulent flow with heat transfer in plane and curved wall jets. *J. Fluid Mech.* 1984, 145, 339–360.

7. Ayech, S. B. H.; Habli, S.; Said, N. M.; Bournot, P.; Palec, G. L. A numerical study of a plane turbulent wall jet in a coflow stream. *J. Hydro-environment Res.* 2016, 12, 16–30.

8. Ayech, S. B. H.; Said, N. M.; Bournot, P.; Palec, G. L. Investigation of a turbulent wall jet in forced convection issuing into a directed coflow stream. *J. Turbul.* 2017, 18, 539–559.

9. Herrero, J.; Grau, F. X.; Gnoffi, J.; Giralt, G. A near wall $k$-$\varepsilon$ formulation for high Prandtl number heat transfer. *Int. J. Heat Mass Transfer* 1991, 34, 711–721.

10. Hoch, J.; Jiji, L. M. Two-dimensional turbulent offset jet-boundary interaction. *J. Fluids Eng.* 1981, 103, 154–161.

11. Hoch, J.; Jiji, L. M. Theoretical and experimental temperature distribution in two-dimensional turbulent jet-boundary interaction. *J. Heat Transfer* 1981, 103, 331–336.

12. Wang, X. K.; Tan, S. K. Experimental investigation of the interaction between a plane wall jet and a parallel offset jet. *Exp. Fluids* 2007, 42, 551–562.

13. Vishnuvardhanarao, E.; Das, M. K. Study of the heat transfer characteristics in turbulent combined wall and offset jet flows. *Int. J. Therm. Sci.* 2009, 48, 1949–1959.

14. Kumar, A. Mean flow and thermal characteristics of a turbulent dual jet consisting of a plane wall jet and a parallel offset jet. *Numer. Heat Transfer, Part A* 2015, 67, 1075–1096.

15. Hnaien, N.; Marzouk, S.; Assia, H. B.; Jay, J. Wall inclination effect in heat transfer characteristics of a combined wall and offset jet flow. *Int. J. Heat Flow Fluid Fl.* 2017, 64, 66–78.

16. Hnaien, N.; Marzouk, S.; Ben Assia, H. B.; Jay, J. CFD investigation on the offset ratio effect on thermal characteristics of a combined wall and offset jet flows. *Heat Mass Transfer* 2017, 53, 2531–2549.

17. Asoudi, A.; Majhoub Said, N. M.; Bournot, H.; Le Palec, G. Comparative study of flow characteristics of a single offset jet and a turbulent dual jet. *Heat Mass Transfer* 2019, 55, 1109–1131.

18. Singh, T. P.; Kumar, A.; Satapathy, A. K. Heat transfer and fluid flow characteristics of a turbulent dual jet impinging on a wavy surface. *J. Therm. Sci. Eng. Appl.* 2020, 12, 041017.

19. Singh, T. P.; Kumar, A.; Satapathy, A. K. Role of a sinusoidal wavy surface in enhancement of heat transfer using turbulent dual jet. *J. Heat Transfer* 2021, 143, 032002.

20. Holland, J. T.; Liburdy, J. A. Measurements of the thermal characteristics of heated offset jets. *Int. J. Heat Mass Transfer* 1990, 33, 69–78.

21. Kim, D. S.; Yoon, S. H.; Lee, D. H.; Kim, K. C. Flow and heat transfer measurements of a wall attaching offset jet. *Int. J. Heat Mass Transfer* 1996, 39, 2907–2913.

22. Tritton, D. J. *Physical Fluid Dynamics*; Von Norstrand Reinhold: UK, 1997; pp 284–286.
(23) Pelfrey, J. R.; Liburdy, J. A. Mean flow characteristics of a turbulent offset jet. *Trans. ASME* 1986, 108, 82–88.

(24) Song, H. B.; Yoon, S. H.; Lee, D. H. Flow and heat transfer characteristics of a two-dimensional oblique wall attaching offset jet. *Int. J. Heat Mass Transfer* 2000, 43, 2395–2404.