Experimental and numerical study of an offset jet with different velocity and offset ratios

Ali Assoudia, Sabra Hablib, Nejla Mahjoub Saïdb, Hervé Bournotc and Georges Le Palecc

aLGM, National Engineering School of Monastir, Mechanical Engineering Department, University of Monastir, Monastir, Tunisia; bLGM, Preparatory Institute for Engineering Studies, Physics Department, University of Monastir, Monastir, Tunisia; cIUSTI, UMR 7343, Faculty of Science, Mechanical Engineering Department, University of Aix-Marseille, Marseilles, France

(Received 1 January 2015; final version received 25 June 2015)

The present study examines the configuration of an offset jet issuing into either a quiescent medium or a moving stream (co-flowing). The mean velocity and turbulence characteristics of the turbulent offset jet have been investigated using a particle image velocimetry technique at three velocity ratios and for two offset ratios. A numerical simulation of a three-dimensional offset jet has also been carried out by comparing the corresponding results with previous experimental data and our measurements. The numerical investigation was performed by means of the finite volume method together with a second-order turbulent closure model – the Reynolds stress model (RSM) – to examine the behavior of the flow for different velocity ratio and offset ratios. Results give a satisfactory agreement between the experimental data and the calculations. Data from the early flow region clearly show a significant influence of the velocity ratio and the offset ratio on the mean flow and turbulence characteristics.

Keywords: offset jet; co-flow stream; particle image velocimetry; numerical simulation; velocity ratio; offset ratio

1. Introduction

Jet flows have been extensively explored in the literature because of their presence in more than one area. These types of flows are either free or bounded, depending upon the distance of the jet boundaries containment. When borders (parallel to the jet axis) are sufficiently far from the output of the jet, it is defined as a free jet. However, a bounded jet occurs when the flow interacts with a parallel wall. Bounded jets can be classified into three families based on the orientation of the containment wall: (a) an impinging jet aimed toward the boundary, (b) a wall jet where the fluid is discharged tangentially to the boundary, and (c) an offset jet, which is formed when a jet discharges into a medium above a horizontal solid wall parallel to the axis of the jet exit but which is offset by a certain distance.

The flow field of an offset jet is complex and has gained extensive research interest, primarily due to their diverse and highly useful practical and engineering applications, such as vertical takeoff and landing (VTOL) aircraft engines, heating, ventilation, and air conditioning (HVAC) systems, and fuel injection systems. The behavior of an offset jet is affected by the presence of a co-flow stream. Indeed, the offset jet is characterized by three main regions (Figure 1). In the first zone, the entrainment of the fluid between the jet, the offset wall and the horizontal wall creates a low pressure zone, forcing the jet to deflect towards the wall and eventually attaching to it at the impingement point. This is called the ‘Coanda effect’.

Within a very short distance from the nozzle exit, the jet flow is dominated by momentum and has the properties of a free jet. In the impingement region, as the jet deflects towards the wall, the flow has features qualitatively closed to other reattaching shear layers flows, like the flow over a backward facing step (BFS). Far away, downstream from the attachment point, the offset jet has the characteristics of a wall jet.

Two-dimensional offset jets have been experimentally investigated by several authors (Bourque & Newman, 1960; Levin & Manion, 1962; Rajaratnam & Subramanya, 1968; Sawyer, 1960, 1963). These authors used various measurement techniques such as hotwires, Pitot tubes and laser Doppler anemometry (LDA) to develop a more general uniform equation that predicts the impingement of the jet to the wall as a function of the offset ratio (the vertical offset distance normalized by the jet nozzle diameter). They also included the measurements of the mean velocity and static pressure distributions.

Most studies on a turbulent offset flow involved jets with large offset ratios \((h/d > 5)\). Hoch and Jiji (1981b) experimentally examined an offset jet in the presence of a free-stream motion using the hotwire technique. They reported measurements of the maximum velocity decay which are in good agreement with results reported by Rajaratnam and Subramanya (1968). For the same previous configuration, Hoch and Jiji presented the analytical solution of the thermal characteristics. They measured

*Corresponding author. Email: nejla.mahjoub@fsm.rnu.tn

© 2015 Taylor & Francis
the temperature distributions of this flow relative to an adiabatic surface, and they found that the temperature within the recirculation zone is uniform and decreases with the increase of the injection height.

Pelfrey and Liburdy (1986a, 1986b) studied the mean and turbulent flow of an offset jet. They employed the LDA for the measurement of the mean velocity profiles at different locations, ratio of curvature to shear strain rate and the entrainment parameter in the recirculation and attachment regions. In this study, the parallel wall was adiabatic and the offset ratio and Reynolds number considered are 7 and 15,000 respectively. They have mentioned how entrainment and local pressure are affected by the jet curvature in the recirculation and impingement regions.

Holland and Liburdy (1990) studied the thermal characteristics of a heated offset jet over an adiabatic impingement surface for different offset ratios and jet exit Grashof numbers. They examined the temperature profile at different downstream locations, the surface temperature distribution in three regions (the recirculation, impingement and developing wall jet regions) and the decay of the maximum temperature. The experimental results showed that the temperature within the recirculation region is close to the impingement location temperature. However, there is evidence that the thermal distribution is dependent on the offset ratio. They also showed that downstream of the impingement region temperature, profiles become locally similar.

There have been fewer investigations of jets with small offset ratios \((h/d < 5)\) (Gao & Ewing, 2007; Lund, 1986; Nasr & Lai, 1998). Lund (1986) dealt with the reattachment length with a wide range of offset ratios, and these results showed that the ratio of the reattachment to the offset ratio changed with the offset distance. Nasr and Lai (1998), in their work, determined the mean velocities and turbulence characteristics of a turbulent plane offset jet with a small offset ratio of 2.125 and a Reynolds number of 11,000 using LDA. A numerical simulation of the problem was also carried out. Three different turbulence models were considered (the standard \(k-\epsilon\) model, the \(k-\epsilon\) model RNG and the second-order turbulence model RSM). The comparison of the corresponding results to the experimental data showed that the standard \(k-\epsilon\) model is more appropriate for the prediction of the turbulent offset jet with small offset ratios. In the study of Gao and Ewing (2007), an experimental investigation was provided to characterize the development of planar jets discharging from a nozzle adjacent to a parallel wall with small offset ratios, of up to 1, and a Reynolds number of 44,000. Their results mentioned that the initial development of the mean flow field could be divided into five regions (three regions associated with the jet attaching to the wall and two associated with the resulting planar wall jet region).

Many investigations are devoted to the study of the heat transfer characteristics of offset jet flows in a turbulent regime. The work of Vishnuvardhanarao and Das (2008a, 2008b, 2009) and Song et al. (2000) examined the characteristics of conjugate heat transfer of a two-dimensional, steady and incompressible offset jet flow. They show that the local Nusselt number is maximum at the reattachment point and reduces with the offset ratio. However, the conjugate heat transfer study involving a laminar flow has received little attention. Kanna and Das (2005, 2008) carried out a steady state heat transfer study for a two-dimensional, laminar and incompressible offset jet flow. The detailed heat transfer results are presented in terms of isotherm contour and Nusselt number for three offset ratios (0.5, 1 and 1.5), with the Reynolds number varying from 200 to 600 and for three Prandtl numbers \((Pr = 0.01, 1\) and 100). The results revealed that the maximum Nusselt number increases as the Reynolds number and Prandtl number increase. Further, in the recirculation region, a significant
effect on the thermal boundary layer was observed for \( Pr = 0.01 \) and 100, whereas for \( Pr = 0.1 \) it was absent. Kumar Raja et al. (2009) numerically investigated the mixed convection flow and heat transfer characteristics in a two-dimensional laminar offset jet issuing parallel to an isothermal flat plate. The goal of their study was to explore the associated heat transfer process in the mixed convection regime in the range of a Reynolds number of 300 to 600 and Grashof number of \( 10^3 \) to \( 10^7 \). Their results show that the reattachment length is strongly dependent on both the Reynolds number and the Grashof number for the range considered.

Y. Yang and Yeh (1994) have numerically studied the flow field of an offset jet with a Reynolds number of 15,000 for offset ratios of 3, 7 and 11. Their results reported that a zero normal gradient boundary condition could be used at the entrainment and exit boundary only when the flow is fully developed near the exit and when the boundaries are sufficiently far away from the region of interest.

Further, a comparative study of the flow field and turbulence characteristics of a turbulent offset jet has been carried out by Rathore and Das (2013). The numerical results obtained from the standard \( k-\varepsilon \) model were compared with those proposed by Launder and Sharma (1974) and Z. Yang and Shih (1993). They observed a similar profile in the wall jet region due to the resemblance of an offset jet with that of a wall jet flow in the wall jet region.

Li, Huai, and Han (2011) reported a numerical investigation for the interaction between a plane wall jet and a parallel offset jet using large eddy simulation (LES). They used the finite-volume method to discretize the governing equations for a Reynolds number of \( 10^4 \). The profiles of the mean streamwise velocity at some locations show a similarity after a certain distance where the two jets are fully merged. The half-width of the jet decreases near the exit due to the presence of the low-pressure zone and then increases as the downstream distance increases. The authors also noticed that the turbulent intensity increases rapidly to the maximum and then decreases along the downstream direction, which indicates that the interaction is strong near the jet exit.

A global observation shows that most studies are two-dimensional. However, the three-dimensional offset jet has received little attention in the literature. Davis and Winarto (1980) and Agelin-Chaab and Tachie (2011) used a circular jet exit, while Nozaki, Hatta, Nakashima, and Matsumura (1979, 1981) utilized a rectangular nozzle jet to study the three-dimensional configuration. The three-dimensional experimental investigation of Agelin-Chaab and Tachie examined the effects of the offset ratio and Reynolds number on the structure of the turbulent offset jet. The use of particle image velocimetry (PIV) measurements has revealed that, in the early region of the flow development, the Reynolds number and the offset ratio have significant effects on the decay of the maximum mean velocity and growth of the shear layer. However, at large downstream distances, the decay and spread rates were found to be nearly independent of the offset ratio. These authors also reported that in the self-similar region, the location of the maximum mean velocity increased nearly linearly with the streamwise distance. For the 3D offset jet from a square nozzle, Nozaki (1983) reported values for the reattachment length \( (x_r) \). He obtained \( x_r/d = 3.8, 6.7, \) and \( 9.9 \) for \( h/d = 1.5, 2.5, \) and \( 3.5 \) respectively.

In the following, we propose to study a turbulent three-dimensional offset jet discharged into a co-flow stream with different velocity ratios and offset ratios. To achieve this purpose, an experimental study is fulfilled using PIV to describe the dynamic and turbulent characteristics of the flow. Then, a numerical simulation is carried out with the Reynolds stress second-order turbulent model (RSM). After validation of the results by the comparison of numerical and experimental data, the characterization of the resulting flow field is examined in order to investigate both the mean velocity and turbulence characteristics of the jet discharging for different velocity ratios and offset ratios.

2. Experimental setup

The experimental measurements were performed in a wind tunnel (Figure 2). The test section was of rectangular dimensions of \( 3000 \times 200 \times 300 \) mm in the longitudinal, vertical and lateral directions, respectively. The top wall of the tunnel was open and the ground was made of wood and covered with plastic to ensure optimal visibility. One of the side walls was made of wood and painted black to prevent possible light reflections. The second side was made of Plexiglas to allow better visibility, and therefore better flow visualizations and data measurements. The turbulence intensity level of the external flow was less than 0.2%. The motor allowed, through a controller, the introduction of a velocity range of about 0 to 16 m/s. The jet was produced by a smooth pipe with an outer diameter of 12 mm and an internal diameter of \( d = 10 \) mm, through which air was injected. The pipe was placed at an adjustable height \( (h) \) of the horizontal wall. The jet was subjected to an external flow at rest or in uniform co-flow of velocity \( u_\infty \). The velocity profiles of the co-flow were established by a hot wire anemometry (Ben Kalifa, Habli, Mahjoub Sâid, Bournot, & Le Palec, 2014). The thickness of the boundary layer of the main flow was about 10 mm. Indeed, the thickness of the boundary layer was defined as the point where the local velocity was 99% of the velocity of the main flow.

To make a quantitative study of the velocity field of a moving fluid, the experimentally tracked values were depicted by means of PIV (Mahjoub Sâid, Mhiri, Le Palec, & Bournot, 2005). This technique provides instantaneous velocity fields over global domains. The PIV technique records the position over time of small tracer particles introduced into the flow to extract the local fluid velocity (Radhouane, Mahjoub Sâid, Mhiri, Le Palec, & Bournot, 2009). In this study, the fluid ejected by the nozzle was
Figure 2. Experimental setup.

seeded with approximately 1 μm diameter glycerin particles (seeding density of approximately 30 particles per ml of pure jet fluid; see Mahjoub Said, Habli, Bournot, & Le Palec, 2012). The main air co-flow was not seeded. The seeding process is fully described in the circuit sketch (see Figure 2). The flowmeter helped to regulate the flow rate of the seeding particles while the decantation reservoir was used to retain eventual contaminants present within the discharged jet. A vane was used to control the flow rate of the discharged offset jet.

The experimental setup of the PIV system was based upon a TSI Incorporated power view system, including a 50 mJ dual YAG laser which produced two flat pulses, the duration of one ranging from $5 \times 10^{-9}$ to $10^{-8}$ s, a Power View 4M high-resolution cross-correlation camera ($2 \times 2$ k resolution, 12 bits), a synchronizer and *Insight* Windows-based software for acquisition, processing and post-processing. This software allows the synchronization of pulsations according to the observed phenomena, and the adjustment of the time step between two images. The time step in the experiment was 70 μs. In order to avoid errors, the velocity vectors were calibrated at 130 μm/pixel and limited to the representation of the velocity field in the regions where the luminance was strong enough.

Laser pulses from an ND-YAG ‘Neodymium-Doped Yttrium Aluminum Garnet’ laser were expanded to form a thin light sheet using a cylindrical lens. The light sheet was brought from the top of the wind tunnel using a 45° mirror. The light scattered from the seeded particles was captured and the resulting scattering signal was collected by an interline transfer Charge-Coupled Device (CCD) camera. The camera was oriented perpendicular to the plane of the sheet. The collected images were transferred to the control computer. A target image was used to provide accurate scaling of the flow. However, in this process, after the vector field is initially processed, filters are applied to the data to eliminate spurious vectors, and the remaining vectors are linearly interpolated. Then the vector field is reprocessed. This permits the use of a cross-correlation PIV algorithm (Hasselbrink, 1999) which eliminates directional
ambiguity and yields improved resolution over single-image autocorrelation techniques. In this way, the in-plane loss of correlation is minimized and therefore the chance of valid vector detection is maximized. Once an ensemble of instantaneous vector fields has been obtained from a set of images, several post-processing calculations are made. Raw data are cast from pixel units into physical coordinates, and turbulence statistics and derived quantities are calculated, using custom software. The final fields were averages performed with 100 successive acquisitions (Knapp, Bertrand, & Mouret, 2003; Mahjoub Saïd et al., 2005; Mahjoub Saïd, Mhiri, Bournot, & Le Palec, 2008).

For each point, experimental uncertainties were estimated as follows: $(v_{\text{max}} - v_{\text{min}})/v_{\text{av}}$, where $v_{\text{max}}$, $v_{\text{min}}$ and $v_{\text{av}}$ are, respectively, the maximum velocity, the minimum velocity and the average velocity measured over the whole process. This uncertainty rate may be accounted for as follows:

- the total number of flow fields used to compute time averages and root mean square (RMS);
- the development of reverse flows and the deformation of the flow within target areas which may deteriorate the algorithm correlations; and
- excessive velocity gradients and out-of-boundary particle motion.

In addition to these factors, problems associated with particle image overlap, non-uniform illumination of the sample volume, reflections of surfaces, particle coalescence, non-uniform flow seeding and image discretization all contribute to the number of correlation anomalies. These problems, although not as serious as velocity gradient and out-of-boundary effects, contribute to sub-pixel bias errors and are often the cause of spurious vectors (Hart, 1998).

The uncertainties of the experiments in the present study and are often the cause of spurious vectors (Hart, 1998).

The velocity profile of the co-flow was measured with an hot wire anemometer together with a high-precision electronic pressure transducer. With the help of an on-line micropressure calibration system, the uncertainty in the co-flow velocity was estimated to be as large as 3% of a reading.

3. Computational setup

3.1. Governing equations

In this work, the incompressible flow is assumed to be steady, three-dimensional and turbulent. Under these assumptions, the governing dimensional equations can be written as:

$$\frac{\partial \overline{u_i}}{\partial x_j} = 0$$

$$\overline{\rho u_i u_j} = -\frac{1}{\rho} \frac{\partial \overline{p}}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \nu \frac{\partial \overline{u_i}}{\partial x_j} - \overline{u_i u_j} \right)$$

The resolution of these equations requires a turbulent closure model capable of properly characterizing the fluctuating functions. Based on our previous work (Mahjoub Saïd, Mhiri, El Golli, Le Palec, & Bournot, 2003), we chose the second-order closure model RSM. In this model, the following equation is solved (Lien & Leschziner, 1994; Roland, 1993):

$$\frac{\rho \overline{u_i u_j}}{\partial x_k} = \frac{\partial}{\partial x_k} \left( \mu \frac{\overline{u_i u_j}}{\partial x_k} \right)$$

$$\frac{\partial}{\partial x_k} \left( \frac{\mu}{\sigma_k} \frac{\partial \overline{u_i u_j}}{\partial x_k} \right) : \text{turbulent diffusion,}$$

$$\varphi_{ij} = \rho \overline{u_i' u_j'} : \text{pressure strain,}$$

$$\varepsilon_{ij} = 2\mu \frac{\partial \overline{u_i u_j}}{\partial x_k} \frac{\partial \overline{u_i u_j}}{\partial x_k} : \text{dissipation rate.}$$

The equations describing the turbulent kinetic energy ($k$) and the dissipation rate of the kinetic energy ($\varepsilon$) associated with the second-order model are defined as follows:

$$\overline{\rho u_i \frac{\partial k}{\partial x_j}} = \frac{\partial}{\partial x_j} \left( \left( \nu + \frac{\nu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right) + \frac{1}{2} P_{ii} - \varepsilon$$

$$\overline{\rho \frac{\partial k}{\partial x_j}} = \frac{\partial}{\partial x_j} \left[ \left( \frac{\nu_t}{\sigma_k} + \nu \right) \frac{\partial k}{\partial x_j} \right] + C_{e1} \frac{\varepsilon}{2} P_{ii} - C_{e2} \frac{\varepsilon^2}{k}$$

where $P_{ii} = 2\nu_t(\partial \overline{u_i}/\partial x_j)^2$ and $\nu_t = C_{\mu} (k^2/\varepsilon)$.

The empirical constants appearing in the above equations are given by the following values:

$$C_{e1} = 1.44, C_{e2} = 1.92, \sigma_k = 0.82 \text{ and } \sigma_\varepsilon = 1.0.$$
is at \( z = 0 \) (Figure 3). The dimensions of the computational domain are chosen so that they do not affect the numerical solutions. After several tests, the dimensions chosen in this study are respectively: \( L_x = 50 \, d \), \( L_y = 20 \, d \), and \( L_z = 20 \, d \) in the streamwise (flow) direction \( x \), normal to the ground wall \( y \) and spanwise direction \( z \). The height of the co-flow is noted as \( H_T \).

In order to close the problem, the incorporation of boundary conditions associated with the above-described system of differential equations is summarized in Table 1:

### 3.3. Numerical simulation

The numerical code uses the finite volume method based on the algorithm SIMPLER proposed by Patankar and Spalding (1972). The differential equations governing the problem are then integrated over each control volume and discretized using an implicit scheme to obtain a system of algebraic equations. The convection terms are interpolated from the values at the center of every face of the control volume, with a second-order upwind scheme, and the discretization of the diffusion terms is made with a second-order finite differences method. The elimination method of Gauss Seidel associated with an under-relaxation technique is used to solve the resulting tridiagonal matrix.

The iterative process is stopped when the convergence criterion is defined as \( |f^{m+1} - f^m| \leq 10^{-5} \) is satisfied. In the previous equation \( f \) stands for velocity components, turbulent kinetic energy and its dissipation rate, while \( m \) is the iterative number.

The resolution of the system of equations needs a fine mesh in zones where interesting interactions take place. In order to describe well any velocity distribution and the turbulent parameters, particularly near and immediately downstream of the jet, we adopted a non-uniform grid system, strongly tightened near the jet nozzle and near walls.

### 4. Results and discussion

#### 4.1. Validation and comparison of the experimental and numerical results

A comparison between the experimental data of Agelin-Chaab and Tachie (2011) and the computational results obtained by a numerical simulation is shown in Figure 4, for an offset ratio \( h/d = 4 \) and a Reynolds number \( \text{Re} = 10,000 \). The experimental results were measured into a stream at rest. This figure presents the variation of the longitudinal velocity component normalized by the local maximum velocity \( \bar{u}_m/u_0 \) as a function of the dimensionless vertical position \( Y^* = (y - y_m)/(y_{0.5} - y_m) \) for different locations \( x/d = 2, x/d = 6, x/d = 20 \) and \( x/d = 40 \) in the symmetry plane \( z = 0 \). Respectively, \( y_m \) and \( y_{0.5} \) denote the location where the streamwise velocity is maximum and where it is half the local maximum velocity. The agreement with both the experimental data and the numerical calculation is acceptable. However, for the longitudinal location \( x/d = 20 \), in the region \( -2 < Y^* < -1 \), we note a slight difference, probably due to the turbulent character of this zone.

The development of the mean flow was examined by studying the decay of \( u_m \). Thus, the maximum velocity decay normalized by the ejection velocity \( u_m/u_0 \) along the dimensionless longitudinal coordinate \( x/d \) in the symmetry plane \( z = 0 \) for an offset ratio \( h/d = 4 \) and a Reynolds number \( \text{Re} = 10,000 \) is presented in Figure 5. The behavior of the maximum velocity decay describes the different regions of the flow. We note the existence of the potential core region followed by the zone of established flow region where the velocity \( u_m/u_0 \) decreases while progressing downstream. We note a satisfying agreement between our numerical results and the experimental data.
(Agelin-Chaab & Tachie, 2011). Hence, the chosen turbulence model predicts the measured data well and is accurate enough for a parametric study.

Before detailing any of the obtained results, we need to compare our PIV measurements with the numerical results obtained by reproducing the same dimensions as the experimental ones.

Figures 6 and 7 show the evolution of the longitudinal velocity component normalized by the maximum velocity ($\bar{u}/u_m$) for two velocity ratios: $R = 1$ ($u_0 = 3 \text{ m.s}^{-1}$, $u_{\infty} = 3 \text{ m.s}^{-1}$) and $R > 1$ ($u_0 = 4 \text{ m.s}^{-1}$, $u_{\infty} = 3 \text{ m.s}^{-1}$) at an offset ratio $h/d = 5.3$. The profiles were presented at $x/d = 4$ and $x/d = 5$ along the vertical distance $y/d$. The confrontation of the calculated and measured results gives
Figure 5. Streamwise distribution of the local maximum mean velocity in the symmetry: $u_0 = 1.43 \text{ m.s}^{-1}$, $h/d = 4$.

According to the above-discussed comparisons, and in accordance with the better description of the experiments with the RSM, we can conclude that our numerical modeling is acceptable and models the experimental configuration of the offset jet issuing within a co-current flow well.

4.2. Experimental results

4.2.1. Results for different velocity ratios

We propose formulating a detailed description of the flow field of a turbulent offset jet issuing into a moving stream (co-flowing) for different velocity ratios $R = u_0/u_\infty$ ($R = \infty$, $R = 1$ and $R > 1$).

4.2.1.1. Flow structure for $R = \infty$ ($u_0 = 3 \text{ m.s}^{-1}$ and $u_\infty = 0 \text{ m.s}^{-1}$). We analyze, in Figure 8, the structure of the flow issued from an offset discharged into a quiescent medium. Figure 8(a) illustrates the flow visualization of the jet for an offset ratio $h/d = 5.3$. This image shows that, for a jet discharging in a stream at rest, the flow at the outlet of the nozzle is predominant and the jet elapses horizontally parallel to the wall for very large distances downstream and behaves as a free jet. In this case, and due to the difference between the jet core and the ambient fluid, a shearing layer is created from the nozzle exit. Downstream of the exit nozzle, we note the formation of vortex structures in the upper and lower boundaries of the jet frontier. Furthermore, we noticed the absence of wake zone vortices located downstream of the nozzle, delimited by the jet flow, the offset wall and the horizontal wall.

In Figure 8(b), streamlines of the flow are presented in the symmetry plane ($z = 0$). This figure shows that the horizontal penetration of the jet is maintained within the domain for large distances downstream of the jet exit. In this region, we can easily see that downstream of the nozzle exit, the velocity ($u_0$) is the local maximum streamwise velocity for a given location $x/d$, just like a single free jet. This figure showed that the region with a high velocity decreases in width downstream of the nozzle exit. The
streamlines close to the wall are slowed and stagnate at surface impact with the attachment point. One of the characteristics of the shear layer is the reattachment length, which was estimated as the distance from the nozzle exit to the location where the jet reattached to the wall. It is noticed that the reattachment length is increased for a jet flow issuing into an ambient flow at rest. The size of the recirculation eddy is increased considerably in this case. In the wall region, the streamlines become parallel to the horizontal wall.

Figure 9 shows the evolution of the normalized mean velocity ($\bar{u}/u_m$) at different longitudinal positions ($x/d = 4$, $x/d = 5$, $x/d = 7$ and $x/d = 9$) in the symmetry plane ($z = 0$). The profiles of the longitudinal velocity adopt a Gaussian shape, nearly symmetrical relative to the axis of the nozzle exit ($y/d = 5.3$). The symmetry of the profiles is more pronounced close to the nozzle exit ($x/d \leq 4$), indicating that the injected flow behaves as a free jet in this region. In addition, this figure reveals that, for the different longitudinal positions, the values of the longitudinal velocity are positive, indicating the absence of the recirculation zone in this region. The advancement in the streamwise direction similarly promotes an overall decrease in the velocities range.

4.2.1.2. Flow structure for $R = 1$ ($u_0 = 3\, \text{m.s}^{-1}$ and $u_\infty = 3\, \text{m.s}^{-1}$). The flow visualization and experimental streamlines of the flow in the symmetry plane ($z = 0\, \text{m}$) for an injection ratio equal to unity and for an offset ratio $h/d = 5.3$ are shown in Figure 10.

Figure 10(a) illustrates the behavior of the flow downstream of the nozzle exit by an instantaneous visualization captured by the CCD camera. When the injection ratio $R$ is equal to the unity, the jet and the outer flow have comparable strengths, giving rise to equivalent interactions, so we can presume the dominance of a jet flow relative to the exterior flow. The jet in this case takes less time to deflect towards the horizontal wall, which leads to a reduction of the location of the impact point. Therefore, the reattachment length becomes smaller compared to the previous case ($u_\infty = 0\, \text{m.s}^{-1}$). This figure also allows a quantitative analysis of the resulting flow field, such as the formation of Kelvin Helmholtz instabilities. This phenomenon occurs when the flow is subjected to shearing between two fluids which slip over one other at different velocities. These instabilities lead to the ripple and the winding within the interface zone until the formation of the vortex to generate the mixing-layer between the two flows. Depending on wind velocity, these patterns turn clockwise or counterclockwise. In fact, the rotational direction depends on the $R$ factor.

We analyze in Figure 10(b) the experimental streamlines for $R = 1$. This figure reveals that, close to the jet exit, many singular streamlines are horizontal and keep following this direction. Thereafter, these streamlines deflect to the horizontal wall in a region near the exit nozzle as a result of the surrounding flow. By examining streamlines in this figure, there are three main zones. In the first zone (the recirculation zone), we noticed the deflection of the jet by the wall and the decrease of the velocity as the jet approaches the horizontal wall. After the jet impacts on the wall, a part of the fluid is deflected upstream from the attachment point into the recirculation zone to form the wake vortices. Herein the velocity is very weak, justifying the weak intensity of the developed vortices. In the attachment region (zone 2), the flow stretches from the impact point to form another flow type. Here, the flow begins to transform into a wall jet flow. In the third and last region (zone 3), downstream of the impact point, the streamlines develop parallel to the horizontal wall to resemble a wall jet flow.
and for an injection ratio greater than the unity $R > 1$ is plotted. This figure reveals that the penetration and mixing of the jet with the outer flow has a significant effect. The entrainment of the ambient fluid in this case generates a shear jet flow leading to the appearance of turbulent structures of Kelvin-Helmholtz instabilities. These structures lead to the formation of a ‘swirling sheet’ on the border between the two fluids. The rotation of this vortex structure is in the anticlockwise direction. The orientation of the vortices is more apparent on the downstream jet leading edge due to its direct confrontation with the outer flow in this region. The jet deflected towards the horizontal wall and attached to it at a position, downstream from the nozzle exit, greater than that found in the previous case ($R = 1$).

Figure 11(b) illustrates the sectional view of flow streamlines for the velocity ratio $R > 1$ in the symmetry plane. We noticed that, immediately downstream of the nozzle exit, the jet flow is weakly influenced by the wind flow, which allows it to rise a little more in the outer flow. Streamlines indicate the motion of the exterior flow fluid. This figure shows three distinct regions. In the area close to the jet exit, the streamlines start horizontally in the ground wall. In this region, the velocity is maximum along the centerline of the nozzle exit until the deflection of the jet. Then, because of the ‘Coanda effect’, these streamlines are
curved towards the horizontal wall and finally to bind with it at the attachment point. Herein, the velocity decreases to a minimum value where the mean streamlines reattach to the wall. At this point, some of streamlines change direction upstream towards the recirculation region (the blue zone) while others continue to develop to resemble a wall jet flow.

4.2.2. Results for different offset ratios
Flow visualization and mean velocity vectors in the symmetry plane ($z = 0$) are shown in Figure 12 for different offset ratios ($h/d = 5.3$ and $h/d = 7.8$) and for a velocity ratio $R > 1$. The flow visualization reveals the existence of vortex structures of Kelvin-Helmholtz instabilities. The rotation direction depends on the value of the velocity ratio $R$. For $R > 1$ these instabilities turn in the counterclockwise direction.

For a higher offset ratio ($h/d = 7.8$), the jet flow is released from the vertical wall attachment, allowing the jet to spread more horizontally. Indeed, the small offset ratio leads to a gradual decrease of the jet wake and a reduction in the size of the recirculation zone. Consequently the standing vortex was found to be very small compared with a higher offset height. In addition, it is observed that the reattachment length of the jet is longer with the increase of the offset ratio. Consequently, the attachment point is located at a further larger position for $h/d = 7.8$ than for $h/d = 5.3$. The mean velocity vectors for two offset ratios indicate that the jet converges towards the horizontal wall as it proceeds downstream from the nozzle ejection, forming a recirculation flow region upstream.
of the reattachment point. Downstream from the recirculation zone, the jet continues its development to resemble a wall jet flow. For an offset ratio \( h/d = 7.8 \), the jet develops and progresses horizontally across the flow for greater distances than for a jet with a small offset ratio \( h/d = 5.3 \) before impingement with the horizontal wall. Thereafter, the recirculation zone induced by the greater offset ratio \( (h/d = 7.8) \) is larger than that given by the small offset ratio \( (h/d = 5.3) \), which causes the remoteness of the impact point of the jet with the horizontal wall. The reduced pressure in the recirculation region is created by the big vortex and the air entrainment into the jet flow. We also noticed that, before the attachment region, the mean velocity for an offset height \( h/d = 5.3 \) decreases more rapidly compared to an offset ratio \( h/d = 7.8 \). Indeed, the jet is released from the effect of the horizontal wall when it is discharged from a nozzle that is far away from the horizontal wall. This figure highlights that the global mean flow configuration is governed mainly by the offset distance, the extension of which defines the dimensions of the recirculating zone. The outer shear layer spreads faster than the inner shear layer, which is limited by the presence of the attachment wall.

4.3. Numerical results
4.3.1. Results for different velocity ratios
The vertical distributions of the longitudinal and vertical velocity components in the symmetry plane \( (z = 0) \) for different longitudinal locations \( (x/d = 4, x/d = 7 \) and \( x/d = 10) \) and for different velocity ratios \( (R = 1, R < 1, R > 1) \) are shown in Figure 13. This figure shows that, downstream of the exit nozzle, the profiles of the longitudinal velocity component indicate negative values in the vicinity of the horizontal wall for different velocity ratios, indicating the presence of a recirculation zone (this is a region characterized by the presence of vortices). Then, the longitudinal velocity increases as a function of the vertical coordinate \( (y/d) \) to achieve the uniform velocity of the surrounding flow prevailing outside the interaction zone.
Figure 12. (a, b) Flow visualization and (c, d) velocity vectors in the symmetry plane: $R > 1 (u_0 = 4 \text{ m s}^{-1} \text{ and } u_\infty = 3 \text{ ms}^{-1})$. 
Figure 13. Profiles of mean longitudinal and normal velocities in the symmetry plane for different velocity ratios and for $h/d = 5.3$. 
In this figure, it can also be seen that, for different velocity ratios, the longitudinal component increases to compensate for the supplying of fluid through the jet, and it increases until a maximum value ($\bar{u}/u_m = 1$) downstream from the jet exit. The examination of the vertical velocity component shows the existence of two peaks—a peak of positive value, located in the recirculation zone ($y/d = 2$) due to the deflection of the jet towards the horizontal wall, and a second peak of negative value, located close to the nozzle axis ($y/d = 5.3$), indicating that the fluid is entrained from the ambient medium towards the horizontal wall.

Figure 14 introduces numerical streamlines in the symmetry plane for different velocity ratios. The streamline contours illustrate the main, recirculation and entrainment flows of the jet. The flow is characterized by a longitudinal variation of the streamline curvature and skewed
Figure 15. Shear stress components $u' u'/u_m^2$ and $v' v'/u_m^2$ in the symmetry plane for different velocity ratios and for $h/d = 5.3$. 
impingement onto the horizontal wall. Streamlines indicate the motion of the surrounding medium. Streamlines entering the domain from the exterior flow boundary, above the jet, appear to bend in the direction of the jet fluid and almost merge with the leading edge boundary of the jet. These streamlines show the deflection of the jet towards the boundary wall and reattach to the surface in the impingement zone, enclosing a region known as the recirculation region. It is noticed that the increase in the velocity ratio reduces the reattachment length and the position of the attachment point. Downstream from the attachment point, these streamlines continue to develop parallel to the horizontal surface to resemble a wall jet flow. This figure also shows that the qualitative behavior of the creation and propagation of the reverse movement (recirculation region) is similar in the experiment and in the calculation.

In Figure 15, we represent the effect of the velocity ratio on the vertical distribution of the longitudinal ($\bar{u}^2/\bar{u}^2_m$) and the normal ($\bar{v}^2/\bar{u}^2_m$) shear stress components in the symmetry plane ($z = 0$) as a function of the vertical coordinate $y/d$, for several positions of $x/d$ (2, 4 and 7) for an offset ratio ($h/d = 5.3$). This figure depicts that the profiles of the turbulent fluctuations have a similar behavior for different velocity ratios. This figure also highlights the presence

![Figure 16. Normal shear stress component $\bar{u}^2/\bar{u}^2_m$ in the symmetry plane for different velocity ratios and for $h/d = 5.3$.](image)
of peaks at the same position but with different fluctuating intensities. These peaks are located near the outlet of the nozzle within the area of interaction of the jet and the co-flow. It is also noticed that the velocity fluctuations for a low velocity ratio ($R < 1$) are stronger compared to those for $R = 1$ and $R > 1$. This is due to the effect of the downwash jet for a lower velocity ejection to the wind flow. We also see that the magnitude of the turbulent fluctuations in the inner shear layer increases as the flow moves downstream, until the jet attaches to the horizontal wall (in the recirculation zone). Thereafter, these turbulent fluctuations decrease near the impact point due to the interaction of the jet flow and the horizontal wall. This result is in good agreement with that found by Gao and Ewing (2007), who reported that the magnitude of turbulent fluctuations in the inner shear layer increases downstream from the attachment point and then decreases due to interaction with the wall.

Figure 16 presents the vertical distribution of the normal shear stress component $\frac{u'v'}{u_m^2}$ in the symmetry plane ($z = 0$) as a function of the vertical coordinate $y/d$, for several longitudinal positions ($x/d = 2$, $x/d = 4$, $x/d = 7$ and $x/d = 10$). Three velocity ratios ($R = 1$, $R < 1$, $R > 1$) are illustrated in this figure for an offset ratio $h/d = 5.3$. This figure shows that the most significant variations occur in the vicinity of the jet outlet. Indeed, the jet flow penetrates

Figure 17. Profiles of the longitudinal mean velocity in the symmetry plane for different offset ratios and for $R > 1$ ($u_0 = 4 \text{ m.s}^{-1}$ and $u_\infty = 3 \text{ m.s}^{-1}$).
Figure 18. Shear stress components $\frac{u'^2}{\bar{u}^2}$ and $\frac{v'^2}{\bar{u}^2}$ in the symmetry plane for different offset ratios and for $R > 1$ ($u_0 = 4 \, \text{m.s}^{-1}$ and $u_\infty = 3 \, \text{m.s}^{-1}$).

within the surrounding flow in this region. These profiles indicate peaks with different values for different velocity ratios, which are located in the recirculation region. Close to the nozzle exit (for $x/d = 2$), there are similar values of these peaks for different velocity ratios. Far downstream from the nozzle ($x/d = 10$), the recorded values are
larger for a velocity ratio less than unity ($R < 1$). Progression through the flow direction (near the attachment point) favors an overall decrease in the shear stress component for different velocity ratios.

### 4.3.2. Results for different offset ratios

Profiles of the mean longitudinal velocity ($u/u_m$), are shown in Figure 17, for different offset ratios ($h/d = 5.3$ and $h/d = 7.8$) and for a velocity ratio $R > 1$ in the symmetry plane ($z = 0$). This figure indicates negative values of the longitudinal velocity related to the presence of a recirculation zone. In this region, values of the longitudinal velocity are higher for $h/d = 7.8$, showing that the size of the recirculation zone is larger for a higher velocity ratio. This indicates that the maximum streamwise velocity of the reverse flow is very large for a higher offset height. The relatively lower maximum reverse flow velocity for the small offset jet is due to the retarding effect of the wall on the recirculating flow. Hence, the reattachment length is higher for a jet emitted with an offset ratio $h/d = 7.8$. Therefore, the position of the impact point of the jet on the horizontal wall is located at a position farther from the nozzle exit.

Figure 18 represents the vertical distribution of the normal $\overline{uu}/u_m^2$ and $\overline{v'v'}/u_m^2$ shear stress components in the symmetry plane ($z = 0$) for different offset ratios ($h/d = 5.3$ and $h/d = 7.8$) and for a velocity ratio $R > 1$. These distributions are plotted in different longitudinal positions ($x/d = 2$, $x/d = 4$ and $x/d = 7$). Profiles of the shear stress show that the distance from the offset of the jet to the impact wall causes significant slowing and suppression effects in the recirculation and attachment regions. It is also noted that these fluctuations have maximum values in the inner shear layer and occur upstream of the impact point. This indicates a strong interaction between the flow in the recirculation region and the inner shear layer near the impact point. It is also noticed that the Reynolds stresses in the internal shear layer close to the attachment point increase with the offset ratio. This figure shows a similarity of the profiles of the longitudinal and vertical shear stress components for the two offset ratios.

Figure 19 presents the wall pressure distribution for two offset ratios ($h/d = 5.3$ and $h/d = 7.8$) tested in the symmetry plane ($z = 0$). For both of the offset ratios, it is noticed that the static pressure decreases in the direction of the flow to a minimum value located close to the center of the recirculation region ($x/d = 10$), indicating the end of the first region when the jet is deflected towards the horizontal wall. The strength of the recirculation region decreases with the offset ratio, as can be seen from the decrease in the magnitude of the minimum pressure. Then, the static pressure increases rapidly through the second region (the impingement region) to a maximum value at a downstream location in the vicinity of the attachment point. In this region, the pressure is generated by the deflection in the streamline after attaching to the horizontal wall, at which point it changes its direction to be parallel to the wall. Figure 19 also highlights that the position of the negative-pressure core and the attachment point move downstream with an increase in the offset ratio. This is probably due to the difference in the trajectory of the jet towards the horizontal wall.

Figure 20 shows the contours of the static pressure for two offset ratios ($h/d = 5.3$ and $h/d = 7.8$) in the symmetry plane ($z = 0$). At the jet exit, and because of the
Figure 20. Contours of the static pressure in the symmetry plane for different offset ratios.

entrainment of the air limited by the jet and the offset wall, we note the creation of a sub-atmospheric pressure region below the jet, forcing the jet to curve towards the boundary wall and impact it at the attachment point, which leads to the formation of three regions (recirculation, impingement, and wall jet region). This phenomenon is known as the Coanda effect. The existence of a negative static pressure zone upstream of the attachment point can be seen in the recirculation region, owing to the existence of the vortex structures. In the impingement region, the pressure is higher than the ambient pressure and the static pressure increases rapidly to a maximum value close to the attachment point.

5. Conclusion

The structure of the flow resulting from the interaction of the offset jet and the exterior flow is very complex. This complexity is generated by the manifestation of various diversified vortices: the vortices established under the jet, the eddies caused by the recirculation zone and the vortices of the shear layer formed in the upper interface jet with the external flow. We examined in this paper the dynamic and turbulent characteristics of a flow issuing from a circular offset jet discharged in a co-flow stream or in a quiescent medium. The flow field was studied for different velocity ratios and offset ratios. This study was conducted both experimentally by means of the PIV technique and numerically by means of the finite volume method, together with a turbulent closure model. A close agreement was achieved between the experimental and numerical results, which show that, in the presence of a co-flow stream, the vertical expansion of the offset jet is limited. It was also observed that the value of the velocity ratio affects the size of the recirculation zone, which begins downstream from the nozzle exit, between the vertical and horizontal walls and the lower border of the jet.

Similarly, we have studied the influence of the offset ratio on the flow. The results indicate that increasing the offset ratio gives a better distribution of the jet within the flow field, giving rise to a better dynamic mixture. It was also found that the reattachment length increases linearly with the offset ratio and the impact of the jet with the horizontal wall is located at a position farther from the offset of the jet. However, for a lower offset ratio, the attachment point is at a closer distance from the nozzle, which implies a spatial reduction of the recirculation zone.
Disclosure statement
No potential conflict of interest was reported by the authors.

References
Agelin-Chaab, M., & Tachie, M. F. (2011). Characteristics and structure of turbulent three-dimensional offset jets and wall jets. *International Journal of Heat and Fluid Flow*, 32(3), 608–620.

Ben Kalifa, R., Habli, S., Mahjoub Saïd, N., Bournot, H., & Le Palec, G. (2014). Numerical and experimental study of a jet in a crossflow for different velocity ratio. *Journal of the Brazilian Society of Mechanical Sciences and Engineering*, 36, 743–762.

Bourque, C., & Newman, B. G. (1960). Reattachment of a two-dimensional incompressible jet to an adjacent flat plate. *Aero Quart.*, 11, 201–232.

Davis, M. R., & Winarto, H. (1980). Jet diffusion from a circular nozzle above a solid plane. *Journal of Fluid Mechanics*, 101, 201–221.

Demuren, A. O., & Rodi, W. (1987). Three-dimensional numerical calculations of flow and plume spreading past cooling towers. *Journal of Heat Transfer*, 109, 113–119.

Gao, N., & Ewing, D. (2007). Experimental investigation of planar offset attaching jets with small offset distances. *Experiments in Fluids*, 42, 941–954.

Hart, D. H. (1998). The elimination of correlation errors in PIV processing. Proc. of the 9th intl symposium on applications of laser techniques to fluid mechanics. Lisbon, Portugal, 8p.

Hasselbrink, E. F. (1999). *Transverse jets and jet flames: structure, scaling and effects of heat release* (Ph. D. Thesis). Stanford University.

Hoch, J., & Jiji, L. M. (1981a). Theoretical and experimental temperature distribution in two dimensional turbulent jet-boundary interaction. *Journal of Heat Transfer*, 103, 331–335.

Hoch, J., & Jiji, L. M. (1981b). Two-dimensional turbulent off-set jet-boundary interaction. *Journal of Fluids Engineering*, 103, 154–161.

Holland, J. T., & Liburdy, J. A. (1990). Measurements of the thermal characteristics of heated offset jet. *International Journal of Heat and Mass Transfer*, 33, 69–78.

Kanna, P. R., & Das, M. K. (2005). Conjugate heat transfer study of two dimensional laminar incompressible offset jet flows. *Numerical Heat Transfer, Part A: Applications*, 48(7), 671–691.

Kanna, P. R., & Das, M. K. (2008). Heat transfer study of two-dimensional laminar incompressible offset jet flows. *International Journal of Thermal Sciences*, 47, 1620–1629.

Knapp, Y., Bertrand, E., & Mourié, F. (2003). 2D-PIV measurements of the pulsatile flow in a left heart simulator. Proceedings of PSVFIP-4, June 3–5, Chamonix, France.

Kumar Rajan, K., Manab, K. D., & Rajesh, K. P. (2009). Numerical study of mixed convection in a two dimensional laminar incompressible offset jet flow. *International Journal of Heat and Mass Transfer*, 52, 1023–1035.

Lauder, B. E., & Sharma, B. I. (1974). Application of the energy dissipation model of turbulence to the calculation of flow near a spinning disc. *International Communications in Heat and Mass Transfer*, 1, 131–137.

Levin, S. G., & Manion, F. M. (1962). Jet attachment distance as a function of adjacent wall offset and angle. Technical report, Harry Diamond Laboratories, Army Materiel Command, Washington DC.

Lien, F. S., & Leschziner, M. A. (1994). Assessment of turbulent transport models including non-linear RNG eddy viscosity formulation and second-moment closure. *Computers and Fluids*, 23, 983–1004.

Li, Z.-W., Huai, W.-X., & Han, J. (2011). Large eddy simulation of the interaction between wall jet and offset jet. *Journal of Hydrodynamics, Series B*, 23(5), 544–553.

Lund, T. S. (1986). Augmented thrust and mass flow associated with two-dimensional jet reattachment. *AIAA Journal*, 24, 1964–1970.

Mahjoub Saïd, N., Habli, S., Bournot, H., & Le Palec, G. (2012). The near field in the mixing of a three-dimensional inclined pollutant jet with a cross flow. *Journal of Enhanced Heat Transfer*, 19(2), 161–178.

Mahjoub Saïd, N., Mhiri, H., Bournot, H., & Le Palec, G. (2008). Experimental and numerical modelling of the three-dimensional incompressible flow behaviour in the near wake of circular cylinders. *Journal of Wind Engineering & Industrial Aerodynamics*, 96(5), 471–502.

Mahjoub Saïd, N., Mhiri, H., El Golli, S., Le Palec, G., & Bournot, P. (2003). Three-dimensional numerical calculations of a jet in an external crossflow: Application of dispersion of pollutants. *Journal of Heat Transfer*, 125, 510–522.

Mahjoub Saïd, N., Mhiri, H., Le Palec, G., & Bournot, P. (2005). Experimental and numerical analysis of pollutant dispersion from a chimney. *Atmospheric Environment*, 39, 1727–1738.

Nasr, A., & Lai, J. (1998). A turbulent plane offset jet with small offset ratio. *Experiments in Fluids*, 24, 47–57.

Nozaki, T. (1983). Reattachment flow issuing from a finite width nozzle (Report 4. Effects of Aspect Ratio of the Nozzle). *Bull JSME*, 26(221), 1884–1890.

Nozaki, T., Hatta, K., Nakashima, M., & Matsumura, H. (1979). Reattachment flow issuing from a finite width nozzle. *Bull JSME*, 22(165), 340–347.

Nozaki, T., Hatta, K., Nakashima, M., & Matsumura, H. (1981). Reattachment flow issuing from a finite width nozzle, (Report 2. Effects of Initial Turbulence Intensity). *Bull JSME*, 24(188), 363–369.

Patankar, S. V., & Spalding, D. B. (1972). A calculation procedure for heat, mass and momentum transfer in three dimensional parabolic flows. *International Journal of Heat and Mass Transfer*, 15, 1787–1806.

Pelfrey, J. R. R., & Liburdy, J. A. (1986a). Effect of curvature on the turbulence of a two-dimensional jet. *Exp Fluids*, 4, 143–149.

Pelfrey, J. R. R., & Liburdy, J. A. (1986b). Mean flow characteristics of a turbulent offset jet. *Journal of Fluids Engineering*, 108, 82–88.

Radhouane, A., Mahjoub Saïd, N., Mhiri, H., Le Palec, G., & Bournot, P. (2009). Impact of the initial streamwiseinclination of a double jet emitted within a cool crossflow on its temperature field and pollutants dispersion. *Heat and Mass Transfer*, 45(6), 805–823.

Rajaratnam, N., & Subramanya, K. (1968). Plane turbulent reattached wall jets. *J. Hydr. Div.*, 94, 95–112.

Rathore, S. K., & Das, M. K. (2013). Comparison of two low-Reynolds number turbulence models for fluid flow study of wall bounded jets. *International Journal of Heat and Mass Transfer*, 61, 365–380.

Roland, S. (1993). *Modélisation Et Simulation Des Ecoulements Turbulents*. Paris: Hermes.
Sawyer, R. A. (1960). The flow due to a two dimensional jet issuing parallel to a flat plate. *Journal of Fluid Mechanics*, 9, 543–561.

Sawyer, R. A. (1963). Two dimensional reattaching jet flows including the effect of curvature on entrainment. *Journal of Fluid Mechanics*, 17, 481–498.

Song, H. B., Soon, H. Y., & Dae, H. L. (2000). Flow and heat transfer characteristics of a two-dimensional oblique wall attaching offset jet. *International Journal of Heat and Mass Transfer*, 43, 2395–2404.

Vishnuvardhanarao, E., & Das, M. K. (2008a). Computation of mean flow and thermal characteristics of incompressible turbulent offset jet flows. *Numer Heat Transf Part A*, 53(8), 843–869.

Vishnuvardhanarao, E., & Das, M. K. (2008b). Study of conjugate heat transfer from a flat plate by turbulent offset jet flow. *Numerical Heat Transfer, Part A: Applications*, 53(5), 524–542.

Vishnuvardhanarao, E., & Das, M. K. (2009). Study of the heat transfer characteristics in turbulent combined wall and offset jet flows. *International Journal of Thermal Sciences*, 48, 1949–1959.

Yang, Y. T., & Yeh, Y. Y. (1994). Numerical study of turbulent offset jets with entrainment boundary. *Journal of Thermophysics and Heat Transfer*, 8(1), 182–184.

Yang, Z., & Shih, T. H. (1993). New time scale based k-ε model for near-wall turbulence. *AIAA Journal*, 31(7), 1191–1198.