Research Article

Numerical Study on Plane and Radial Wall Jets to Validate the 2D Assumption for an Idealized Downburst Outflow

Yongli Zhong^{1,2}, Zhitao Yan^{1,2}, Yan Li¹, Jie Luo³, and Hua Zhang⁴

¹School of Civil Engineering and Architecture, Chongqing University of Science and Technology, Chongqing 401331, China

²Chongqing Key Laboratory of Energy Engineering Mechanics & Disaster Prevention and Mitigation, Chongqing 401331, China

³China Construction Fourth Engineering Bureau Guizhou Investment and Construction Co., Ltd., Guiyang 550014, China

⁴School of Civil Engineering, Chongqing University, Chongqing 400045, China

Correspondence should be addressed to Zhitao Yan; zhitaoyan@qq.com

Received 18 March 2021; Revised 2 October 2021; Accepted 18 October 2021; Published 10 November 2021

Academic Editor: Gang Zhou

Copyright © 2021 Yongli Zhong et al. This is an open access article distributed under the Creative Commons Attribution License, which permits unrestricted use, distribution, and reproduction in any medium, provided the original work is properly cited.

Turbulent radial and plane wall jets have been extensively investigated both experimentally and numerically over the past few decades. Previous studies mostly focused on the heat and mass transfers involved in jet flows. In this study, a comprehensive investigation was conducted on turbulent radial and plane wall jets, considering both jet spread and velocity decay for different parameters. The numerical results were compared with existing experimental measurements. The comparison focused on the velocity profile, jet spread, and velocity decay, and revealed that the Reynolds stress model (RSM) performs well in the simulation of both radial and plane wall jets. The results show that with a typical ratio of cloud base height to diameter for most downburst events, the effects of nozzle height and Reynolds number on the evolution of the radial wall jet are not significant. Both the jet spread and velocity decay exhibit a clear dependence on the Reynolds number below a critical value. Above this critical value, the plane wall jet becomes asymptotically independent of the Reynolds number. The co-flow was found to have a significant influence on the evolution of the plane wall jet. Comparatively, the jet spread and velocity of the radial wall jet were faster than those of the plane jet. For applications in civil engineering, it is valid to approximate the downburst outflow with a two-dimensional (2D) assumption from the perspective of longitudinal evolution of the flows.

1. Introduction

The occurrence of wall jet flows is common in many industrial applications. Traditional applications include ventilation, film cooling, and separation control over wings [1]. In some small-scale geometric engineering applications, turbulent jets are mainly used for heat transfer [2, 3]. In civil engineering, the wall jet flow can be used to design laboratory (or numerical) simulations of large-scale downburst outflow, which is a high-intensity wind that results in failures of transmission lines [4, 5]. There are two main methods for generating wall jet flows depending on the angle of injection of the high-momentum fluid. If the high-momentum fluid is injected normally to the wall, the resulting flow field is classified as a radial wall jet [6], which is a logical way to achieve flow similarity of downburst outflow [7]. A

plane wall jet is produced when the high-momentum fluid is parallel to the wall, and this can also be an idealized model for the outflow region of a downburst [8, 9].

Both radial and plane wall jets have been extensively studied over the past few decades. Owing to the extensive applications of wall jets, there are many studies on them. Launder and Rodi [10] provided a comprehensive review that reflects the state-of-the-art experimental research conducted until 1980. Reviews on more recent literature can be found in Naqavi et al. [11] and van Hout et al. [12] for plane and radial wall jets, respectively. These literature reviews suggest that the studies on wall jets usually focus on one type at a time. Radial [13–15] and plane [16–20] wall jets were investigated separately. Only a few studies have compared the basic characteristics, such as the evolution of the length and velocity scales, of the two types of wall jets.

Tanaka and Tanaka [21] compared the velocity and length scales of their experimental radial wall jet with those available in the literature on plane wall jets. It was revealed that the evolution of the half-maximum velocity location is very similar, while there is a difference between the evolutions of the maximum velocities. Banyassady and Piomelli [22] obtained similar conclusions through a large eddy simulation. Guo et al. [23] identified that the confinement of an impinging jet has no significant effect on the velocity decay rate, and the presence of the upper confinement plate accelerates the wall jet growth rates compared to those reported in the previous plane and radial wall jet experiments. However, the numerical study results of radial wall jet by Fillingham and Novosselov [24] exhibited an excellent agreement with those of plane wall jet reported by Naqavi et al. [11] in terms of the evolution of both length and velocity scales. Bagherzadeh et al. [2] reported that wall roughness influences the decay rate. They found that the decay of velocity increases with an increase in wall roughness.

Most previous studies on plane and radial wall jets focused on heat and mass transfers; they had relatively small Reynolds numbers [25, 26] and were not suitable for the study of downbursts. Due to the influence of the wall, the free jet region of the impinging jets is also different from the turbulent round jet [27]. In applications related to civil engineering, the majority of the previous investigations on radial and plane wall jets focused on the profiles and time series of velocity [28–30]. It is useful to characterize the length and velocity scales for high Reynolds numbers. In addition, an external stream exists in most practical situations of a plane wall jet. The external stream also provides fluid for jet entrainment [31]. In the case of downburst outflow simulation, an external stream may be applied to simulate a translating event [32]. In addition, the results from different wall jet studies have apparent discrepancies even in some basic characteristics. There is a fundamental geometric difference between plane and radial wall jets: radial wall jets have an additional direction of expansion. Although Lin et al. [33] verified that the frontal curvature has little effect on the resultant wind loading of a structure within a certain transverse width, which is a geometric analysis, the validity of the 2D assumption needs to be further investigated from the perspective of the difference between the longitudinal evolution of the 3D outflow and 2D wall jet.

The primary objectives of the present study were (1) to perform a systematic parametric study through the numerical simulation of radial and plane wall jets to determine the characteristics of evolution and (2) to further investigate the 2D assumption for the downburst outflow. Following Section 1, Section 2 introduces the numerical simulation details of the radial and plane wall jets; Section 3 presents the comparison of the results from existing literatures with those predicted in the current simulation. The Reynolds-number dependency was investigated for both types of wall jets. Subsequently, the influence of the nozzle height from the plate on the radial wall jet and the effect of co-flow on the plane wall jet are presented. Based on the simulation results,

the accuracy of approximating a downburst outflow with a plane wall jet was evaluated, as presented in Section 4. Section 5 summarizes the main findings of the present study.

2. Problem Formulation

2.1. Governing Equations. A commercially available computational fluid dynamics (CFD) package, FLUENT, was used to simulate the impinging jet and plane wall jet. The conservation equations of mass and momentum for an incompressible fluid flow can be expressed as follows:

$$\begin{aligned} \frac{\partial u_i}{\partial x_i} &= 0, \\ \frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) &= -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j} + \frac{\partial}{\partial x_j} \left(\mu \frac{\partial u_i}{\partial x_j} \right), \end{aligned} \quad (1)$$

where ρ is the fluid density; u_i and u_j are the mean velocity components corresponding to i and j , respectively; p is the pressure; μ is the fluid viscosity; and t is the time. The Reynolds stress tensor, $\tau_{ij} = -\rho u_i' u_j'$, needs to be numerically modeled to close the equations.

An exhaustive investigation on a plane wall jet was conducted by Yan et al. [34] using seven Reynolds-averaged Navier–Stokes (RANS) turbulence models and a large eddy simulation (LES). They found that the stress-omega Reynolds stress model (SWRSM) with adjusted turbulence model constants achieved the best results in simulating a steady wall jet without co-flow. In addition, Sengupta and Sarkar [35] indicated that the LES model, realizable k - ϵ model, and Reynolds stress model (RSM) perform better in simulating an impinging jet. In the current study, the Navier–Stokes equations were closed by employing RSM (stress-omega). The stress-omega model is a stress-transport model proposed by Wilcox [36], and a revised version was introduced subsequently [37]. The default constants from the original version [36] are used in FLUENT 16.0. The revised version was used in the current study. The parameters of the SWRSM are listed in Table 1.

2.2. Assumptions and Numerical Solution. The assumptions made for solving the pressure and flow fields inside the wind channel and their corresponding implications are as follows:

- (1) Constant and uniform properties, i.e., ρ and μ are constant.
- (2) Newtonian fluid.
- (3) Isotropic fluid.
- (4) Isothermal fluid.
- (5) Incompressible fluid.
- (6) Stokes' hypothesis holds true.

The differential equations governing the flow were integrated using the finite-volume method, which is a specific case of residual weighting methods [38, 39]. The least squares cell-based method was adopted for the numerical approximation of gradients, and bounded central

TABLE 1: Coefficients for SWRSM.

C_1	C_2	$\text{Alpha}^*_{\text{inf}}$	$\text{Alpha}_{\text{inf}}$	Beta_i	$\text{Beta}^*_{\text{inf}}$	zeta^*	Mt0	TKE Prandtl number	SDR Prandtl number
1.8	10/19	1	0.52	0.0708	0.09	0.5	0.25	5/3	2

differencing was selected for the momentum discretization. The pressure was discretized using a second-order implicit scheme. The discretized governing equations were solved using the Semi-Implicit Method for Pressure-Linked Equations Consistent (SIMPLEC) algorithm, which is an improved version of the SIMPLE algorithm [40], for pressure–velocity coupling. The SIMPLEC algorithm adopted in this study is presented in Appendix A. The time step was selected to guarantee that the Courant number was less than one to maintain the stability of the computation. The absolute convergence criteria for the continuity equation and the other equations are 1×10^{-6} and 1×10^{-5} , respectively.

2.3. Flow Configuration and Computational Setup. The computational domains for the plane wall jet and impinging jet using the Cartesian coordinate system are shown in Figure 1, where b is the jet inlet height of the plane wall jet and D is the circular inlet diameter. For the plane wall jet domain, Yan et al. [41] indicated that the development of the wall jet is not affected up to the streamwise position $x/b = 80$ for $h/b = 20$, where h is the co-flow height. The domain height of the plane wall jet assumed in this study was $21b$. The bottom was set as a wall boundary with a no-slip condition. At the inflow plane, a velocity profile was set for the wall jet up to $y/b = 1.0$, and the rest of the plane had a uniform co-flow, U_E . Jet entrainment was provided by a uniform co-flow. The top boundary was specified as a free-slip boundary condition. The spanwise direction had periodic boundaries to attain two dimensionalities, and at the exit plane, a pressure outlet boundary condition was applied, as shown in Figure 1(a). Sengupta and Sarkar [35] showed that the geometric conditions of the domain have little influence on the flow profiles. For the current impinging jet simulation, a three-dimensional (3D) cylindrical domain was used, as illustrated in Figure 1(b). Pressure outlet boundary conditions were applied with a zero-normal gradient at the outflow boundaries. A no-slip condition was assumed at the bottom wall of the computational domain. The turbulence intensity of the inflow was set as 0.01 for both the impinging jet and plane wall jet.

A nonuniform hexahedral grid was used. The nearest node to the wall in the y -direction was located at $y^+ < 1$ for all grids, where $y^+ = \Delta y \rho u_\tau / \mu$ is the nondimensional wall distance, $u_\tau = (\tau_w / \rho)^{1/2}$ is the friction velocity, τ_w is the wall shear stress, μ is the dynamic viscosity of the fluid, and ρ is the density of the fluid. To ensure grid independency, two grids were employed for the radial wall jet simulation: coarse G1 (2.4 million) and fine G2 (4.1 million). Two grids were also employed for the plane wall jet simulation: coarse G3 (1.8 million) and fine G4 (3.2 million). As shown in Figure 2, the results for the two levels of grid resolution were very similar, and there was no noticeable difference between the mean velocities of the radial and plane wall jets. All the

results presented in this work are for the two fine grids shown in Figure 1.

The mean velocity is usually scaled with length and velocity variables, namely, the maximum velocity U_m and the distance $y_{1/2}$ from the wall to the position at which the mean velocity declines to half of its maximum value [7, 42], to obtain self-similar characteristics. Researchers often selected the height y_m of the maximum velocity location as the length scale [43–45] for the vertical mean velocity profile of the downburst.

Hjelmfelt [46] noted that approximately 50% of the observations from the Joint Airport Weather Studies (JAWS) Project were for traveling events. Storm translational velocities can be as high as 1/3 of the downdraft velocity. A downburst with “surface environmental wind” or a downburst embedded in a translating storm can be modeled through a plane wall jet approach with a co-flow [32, 33]. The velocity ratios are defined as $\beta = U_E / U_j$ for the plane wall jet. The Reynolds number is defined as $\text{Re} = U_{\text{jet}} D / \nu$ for impinging jet and $\text{Re} = U_{\text{jet}} b / \nu$ for plane wall jet, where ν is the kinematic viscosity. The simulation cases for investigating the influences of the Reynolds number and nozzle height on the impinging jet are listed in Table 2, while those for investigating the influences of the Reynolds number and co-flow on the plane wall jet are listed in Table 3.

2.4. Numerical Procedure Validation. To verify the reliability and accuracy of the current simulation, the results obtained from the current model were compared with the experimental results obtained by McIntyre [47] for a Reynolds number of $\text{Re} = 30,700$ and a velocity ratio of $\beta = 0.1$. Because the main objective of this study was to characterize the length and velocity scales, normalized mean velocities were used for validation. Figure 3 shows the velocity profiles at two downstream locations. As demonstrated in this figure, there is a good agreement between the current numerical results and experimental data from McIntyre [47]. It was concluded that the present numerical method is valid and can be used to predict the mean flow properties of wall jets.

3. Results and Discussions

3.1. Statistical Properties

3.1.1. Mean Axial Velocity of Impinging Jet. The axial velocity profile along the centerline of the jet at half of the nozzle height from the ground plane is shown in Figure 4. In order to obtain the nondimensional velocity profiles, the mean axial velocity of the impinging jet is normalized by the local maximum velocity V_m , and the radial distance is normalized by the local jet half-width $\delta_{1/2}$ (defined as the width at which the mean axial velocity has decreased to half of its maximum value). L is the downstream distance from the jet nozzle. It can be seen that the current results of

FIGURE 1: Computational domains and grids for radial and plane wall jets. (a) Radial wall jet. (b) Plane wall jet.

FIGURE 2: Grid independence study for: (a) radial wall jet, $r = 1.5D$ and (b) plane wall jet, $x = 30b$.

TABLE 2: Simulation cases for radial wall jet.

Cases	R1	R2	R3	R4	R5	R6	R7	R8	R9	R10
Re ($\times 10^4$)	5	10	15	20	25	5	5	5	5	5
H/D	2	2	2	2	2	1	3	4	5	6

TABLE 3: Simulation cases for plane wall jet.

Cases	P1	P2	P3	P4	P5	P6	P7	P8	P9	P10
Re ($\times 10^4$)	1	2	4	6	8	10	6	6	6	6
β	0.1	0.1	0.1	0.1	0.1	0.1	0.15	0.2	0.25	0.3

impinging jet match well with the experiment of Sengupta and Sarkar [35] on the right side and there are some differences on the left side. This discrepancy may be due to the concentration of seeding particles along the jet boundary,

which is a common problem in the use of PIV. Compared with the velocity profile near the nozzle of a round free jet [48], the profile of impinging jet has a top-hat shape due to the existence of the plate. After impacting the plate, the axial velocity of the impinging jet is transformed into the radial velocity, while the round free jet gradually develops and becomes self-similar [27].

3.1.2. Vertical Profile of Mean Streamwise Velocity in Wall Jet Region. Figure 5 shows the profiles of the mean velocity normalized using U_m and $y_{1/2}$ at $x = 1.5D$ for the radial wall jet with $Re = 50,000$ and $H = 2D$ and at $x = 30b$ for the plane wall jet with $Re = 60,000$ and $\beta = 0.1$. The current numerical results were compared with those of the plane wall jet experiment by Eriksson et al. [17]; the radial wall jet experiment by Cooper et al. [13]; and three empirical models for the vertical profile of the downburst [7, 44, 49]. The current

FIGURE 3: Comparison of mean streamwise velocity profiles, (a) $x = 20b$, (b) $x = 80b$: current simulation (—); McIntyre [47] ($\diamond$).

FIGURE 4: Comparison of normalized axial velocity profiles.

results of the plane wall jet are in good agreement with the vertical profile suggested by Wood et al. [7] and the results obtained by Eriksson et al. [17] for the outer region ($y > y_m$).

However, the current model underestimated the height of maximum velocity (y_m) by 7.8 and 4.9% and overestimated the velocity by 9.1 and 5.2% at the height of $y/y_{1/2} = 0.08$, compared with those obtained by Eriksson et al. [17] and Wood et al. [7], respectively. The current results of the radial wall jet exhibit a larger y_m and agree well with the vertical profile suggested by Oseguera and Bowles [49] for the region of $y/y_{1/2} > y_m$. In the inner region, the current model predicted a velocity lower by 10.1 and 11.8% compared to the experimental results of Cooper et al. [13] and Wood's profile at the height of $y/y_{1/2} = 0.08$, respectively. It can be concluded that both the approaches can generate a flow that is similar to a downburst outflow and are effective in investigating downburst outflows.

3.1.3. Turbulent Quantities. Figure 6 shows the distribution of RMS (root mean square) fluctuations in streamwise velocity profiles at different streamwise locations ($2 < x/b < 3.5$) and radial locations ($40 < r/D < 70$). It can be seen that the results from both wall jet and impinging jet are in good agreement with the literature data in the outer layer. The streamwise RMS velocity profiles of the wall jet show obvious twin-peak behavior and self-similarity, while there is no obvious peak near the wall in the simulation and experimental data of the impinging jet.

FIGURE 5: Vertical mean velocity profiles in outer scales. (a) Radial wall jet. (b) Plane wall jet.

FIGURE 6: Profiles of streamwise RMS velocity. (a) Radial wall jet. (b) Plane wall jet.

Figure 7 presents the RMS profiles of vertical velocity at different streamwise locations (x/b) and radial locations (r/D). The numerical results of the plane wall jet are in good agreement with the experimental data and show obvious self-similarity, while the comparison for impinging jets with the hot-wire data sets is less good. The numerical results of the impinging jet are close to the experimental data of Knowles and Myszko [14] but smaller than the experimental results of van Hout *et al.* [12].

Figure 8 shows the profiles of Reynolds shear stress along the wall-normal direction at different streamwise locations (x/b) and radial locations (r/D). The profiles of Re shear stress of both plane wall jet and impinging jet also have two peak values. Different from streamwise RMS velocity, the peak values of Re shear stress near the wall are negative. The results of the plane wall jet match well with the experimental data of Rostamy *et al.* [18] while the result from Eriksson *et al.* [17] is smaller than the current simulation. The

FIGURE 7: Profiles of vertical RMS velocity. (a) Radial wall jet. (b) Plane wall jet.

FIGURE 8: Profiles of Reynolds shear stress. (a) Radial wall jet. (b) Plane wall jet.

impinging jets values measured by Knowles and Myszko [14] are also considerably lower.

3.2. Jet Spread and Velocity Decay in Plane Wall Jet

3.2.1. Effect of Jet Reynolds Number. The dependence of the flow on the Reynolds number was studied for a fixed velocity ratio ($\beta = 0.1$). The decay of the spread of the jet flow and the

maximum velocity was investigated. The average spread rate of the plane wall jet was found to be linear and can be expressed as the half-height ($y_{1/2}$) with respect to the downstream distance [10]. The growth is represented by equation (2) as follows:

$$\frac{y_{(1/2)}}{b} = A_p \left(\frac{x}{b} \right) + C. \quad (2)$$

Many previous studies have reported the values of the slope A_p . Launder and Rodi [10] summarized a large number of experiments and found that most values of A_p fall within the range of 0.073 ± 0.002 , except those obtained in low-Reynolds-number tests. The experiments of Eriksson et al. [17], where measurements were performed using LDV, indicated that the spread rate should be 0.078 for $Re = 10,000$. The experimental data from Abrahamsson et al. [16] indicated a dependence of the slope on Re and reported that values of A_p varied from 0.075 to 0.081 with $Re = 10,000$ –20,000. Wygnanski et al. [19] also found a clear Reynolds-number dependence and a larger slope value of 0.088 for a Reynolds number ranging from 3,700 to 19,000. With an external stream, Zhou and Wygnanski [20] indicated that the influence of the Reynolds number is less significant when the velocity ratio is large.

Figure 9 shows the streamwise growth of the jet half-height for different Re values with a co-flow ratio of 0.1. The values of the slopes varied from 0.0781 to 0.0733 with $Re = 20,000$ –100,000. The obtained results and the experimental data from Abrahamsson et al. [16] are in good agreement for $Re = 20,000$. The Reynolds number does not seem to have a significant effect on the spread rates in the current study. The decrease in the spread rate tended to gradually decrease as the Reynolds number increased. A similar result was also observed by Abrahamsson et al. [16]. Schwarz and Cosart [50] reported that the variation in the spread rate was not apparent in their study for higher Re numbers ranging from 13,510 to 41,600. Therefore, there should exist a threshold, which was found to be 60,000 in the current study. When the Reynolds number is greater than this threshold, the dependence of the spread rate on the Reynolds number can be ignored. When the wall jet approach is used to simulate the downburst outflow in the boundary layer wind tunnel, the Reynolds number is usually greater than this threshold. The large-scale features of the simulated outflow were independent of the Reynolds number.

Few studies have been conducted on the maximum velocity height (y_m). Zhou and Wygnanski [20] reported that y_m exhibits an approximately linear relationship with the downstream distance, and the Reynolds number has no significant effect on y_m . However, Reynolds-number dependence is observed in the CFD results obtained by Ben et al. [51] who found that y_m decreases linearly with x at a higher rate for a lower Reynolds number. Figure 10 shows the longitudinal distributions of y_m for different Re numbers. The results from the current study are in agreement with the observations by Zhou and Wygnanski [20]. The growth rate of the maximum velocity height remained constant at $dy_m/dx = 0.0133$, whereas in the experiment by Zhou and Wygnanski [20], the value was $dy_m/dx = 0.0114$.

In previous studies [11, 19], the decay of the maximum velocity is represented by the following equation:

$$\frac{U_m}{U_j} = B_{p1} \left(\frac{x - x_0}{b} \right)^{N_p}, \quad (3)$$

FIGURE 9: Plane wall jet growth rate for various Reynolds numbers.

FIGURE 10: Rate of spread of y_m for various Reynolds numbers.

where x_0 is the virtual origin of the wall jet. The virtual origin is used to make the lines fit to converge the data to $U_m/U_j = 1$ at $x = x_0$ [19]. Velocity decay is well documented for the plane wall jet; however, the values of the coefficients B_p and N_p are different in different investigations. The exponents of the power law are reported to be $N_p = -0.47$, -0.56 , and -0.48 by Wygnanski et al. [19], Schneider and Goldstein

[52], and Tang et al. [26], respectively. However, Wygnanski et al. [19] also suggested that their results fit the power law fairly well, with $N_p = 0.5$. Lin [53] found that the arithmetic mean values of N_p and B_p in previous studies were -0.52 and 4.19 , respectively. Barenblatt et al. [54] indicated that $N_p = 0.5$ is necessary for achieving a completely similar flow. In the current study, the power law with $N_p = 0.5$ was used to fit the results. Thus, equation (4) can be rewritten as:

$$\left(\frac{U_j}{U_m}\right)^2 = B_p \left(\frac{x - x_0}{b}\right) + 1. \quad (4)$$

The effect of the Reynolds number on the decay of the maximum velocity $(U_j/U_m)^2$ is shown in Figure 11. The values of B_p decreased with increasing Reynolds number and varied from 0.081 to 0.072 with $Re = 10,000$ – $100,000$. The variation was approximately 13%. When the Reynolds number was higher ($Re > 20,000$), the variation in the coefficient B_p was insignificant. For example, the value of B_p varied from 0.074 to 0.072 with $Re = 60,000$ – $100,000$. The variation was only approximately 2%. This indicates that the effect of the Reynolds number is negligible for higher values ($Re > 60,000$). This is in complete agreement with the observations made by Schwarz and Cosart [50].

3.2.2. Effect of Velocity Ratio. The effect of the velocity ratio on the streamwise development of the half-width and maximum mean velocity for a fixed inlet Reynolds number was investigated. The Reynolds number has no significant effect on the rate of spread and the decay of the maximum velocity for $Re > 60,000$, as explained in Section 3.3.1. Thus, the simulations of the plane wall jet with various velocity ratios were conducted for $Re = 60,000$. The effect of the velocity ratio on the spread rate at half-height is shown in Figure 12. The value of A_p was between the measured values from Zhou and Wygnanski [20] and McIntyre et al. [55], at a velocity ratio of 0.1 . It can be observed that the velocity ratios have a significant effect on the half-height, which is in agreement with the findings by Zhou and Wygnanski [20]. The influence on the half-height, reported in Zhou and Wygnanski [20], is larger, which may be due to the low Reynolds number. The value of A_p decreased as the velocity ratio increased, and the intercepts from various co-flow ratios were nearly the same. However, the velocity ratios also had no significant effect on y_m , and dy_m/dx was still 0.0133 , as shown in Figure 13.

Few studies have reported the values of B_p for different velocity ratios. To compare the decay of the maximum velocity downstream, the studies of Wygnanski et al. [19] and McIntyre et al. [55] were considered. The effect of the velocity ratio on the streamwise decay of the maximum velocity is shown in Figure 14. The result from the current study at $\beta = 0.1$ is similar to that of Wygnanski et al. [19]. McIntyre et al. [55] reported the coefficient $B_p = 0.052$, which is 30% smaller than that in the current study. It was observed

FIGURE 11: Decay of maximum velocity for various Reynolds numbers.

FIGURE 12: Streamwise development of $y_{1/2}$ for various velocity ratios.

FIGURE 13: Rate of spread of y_m for various velocity ratios.

FIGURE 14: Decay of maximum velocity for various velocity ratios.

that the values of B_p decreased as the velocity ratio increased, that is, the decay of the maximum velocity slowed down.

The values of A_p and B_p obtained from the current data for different velocity ratios are shown in Figure 15. It can be observed that the velocity ratio had a significant influence on the evolution of the plane wall jet. The values of coefficient A_p increased linearly with the velocity ratio β , while the coefficient B_p decreased exponentially with an increase in the co-flow ratio β .

FIGURE 15: Values of A_p and B_p with respect to various velocity ratios.

According to the above analyses, the effect of a high Reynolds number can be neglected. Thus, only the effect of the velocity ratio needs to be considered when the plane wall jet method is used to simulate the downburst outflow. The spread of the plane wall jet with co-flow can be expressed as equation (5) and the decay of the maximum velocity with downstream distance for different velocity ratios can be expressed as equation (6).

$$\frac{y_{(1/2)}}{b} = (-0.108\beta + 0.0876) \times \left(\frac{x}{b}\right) + C_1, \quad (5)$$

$$\left(\frac{U_j}{U_m}\right)^2 = 0.11 \exp^{-3.7\beta} \left(\frac{x}{b}\right) + 1. \quad (6)$$

3.3. Jet Spread and Velocity Decay of Impinging Jet

3.3.1. Effect of Jet Reynolds Number. This section explains the effects of the Reynolds number on the jet spread and velocity decay for the wall jet region of the impinging jet. According to Hjelmfelt [46] summary of JAWS results, the average downburst diameter is approximately 1.8 km and the average distance from the cloud base to the ground surface is 2.7 km. On average, the ratio of the cloud base height to the downburst diameter is approximately 1.5. Thus, a widely used distance of $H = 2D$ from the jet nozzle to the bottom wall [56] was adopted in the current study.

For a fixed nozzle height above the plate board, $H = 2D$, the variation in the half-height with r positions for different Reynolds numbers is shown in Figure 16. It can be observed that the Reynolds number has no significant effect on the half-height, which can be considered independent of the Reynolds number. Sengupta and Sarkar [35] and Li et al. [56] proposed empirical expressions for the distribution of half-height based on experimental and CFD data, respectively.

FIGURE 16: Radial growth of jet half-height for different Reynolds numbers.

The current CFD results are very similar to the empirical curve obtained by Sengupta and Sarkar as well as the hot-wire data from Cooper et al. [13] in the region of $r < 3.5D$. The empirical curve obtained by Li et al. [56] and the experimental data reported by Knowles and Myszko [14] exhibit larger values than those in this study. By fitting with the CFD results, the spread of the jet flow can be expressed as

$$\frac{y_{(1/2)}}{D} = -0.1771 + 0.1252 \frac{r}{D} + 0.9418 \exp\left(-1.532 \frac{r}{D}\right). \quad (7)$$

Because of the high convective heat transfer from the wall near the stagnation point, most of the studies on radial wall jets are limited to the stagnation region. The wall jet region of the impinging jet has been less emphasized. Xu and Hangan [45] suggested that the impingement region extended from the free jet axis to the location of $r/D = 1.4$. Tummers et al. [57] reported that the minimum value of half-height for an impinging jet is located at $r/D = 1.5$. Cooper et al. [13] indicated that the radial wall jet grows linearly with distance $r > 2D$, and the nozzle height has little effect on the slope of the jet growth. The results from Knowles and Myszko [14] exhibited a linear growth for $r > 2.5D$. Figure 17 shows the plot of the half-height with r for different Reynolds numbers and $r > 1.8D$. The slope obtained from the current study was 0.098, which is equal to the value reported by van Hout et al. [12].

The effect of the velocity ratio on the radial evolution of the maximum velocity decay, together with the available experimental results, is presented in Figure 18. The

FIGURE 17: Variation of the growth rates for different Reynolds numbers.

FIGURE 18: Decay of maximum mean radial velocity for different Reynolds numbers.

measured results from different studies do not agree well with each other. It can be observed that the values of U_m/U_j for different Reynolds numbers in the same radial position are the same. The maximum mean velocity scaled with the slot quantities is independent of the Reynolds number. This is in agreement with the observations made by Xu and Hangan [45]. Sengupta and Sarkar [35] proposed an empirical expression as follows:

$$\frac{U_m}{U_j} = \exp\left(a - \frac{b}{r/D} - c \ln\left(\frac{r}{D}\right)\right), \quad (8)$$

where $a = 1.905$, $b = 1.858$, and $c = 1.949$. However, to fit with the CFD results of the current simulation, the values of a , b , and c should be adjusted to 2.617, 2.637, and 2.27, respectively. For $1 < r/D < 2$, the current results agree well with those of Knowles and Myszko [14] for $Re = 90,000$. For $r/D > 2$, a good agreement can be observed between the current data and the results obtained by Xu and Hangan [45] for $Re = 43,000$. The maximum radial velocity (U_m) was almost equal to the jet velocity at the radial station for $r/D = 1.1$. This is in agreement with the findings by Tummers et al. [57].

3.3.2. Effect of Nozzle Height. To examine the effect of the nozzle height above the plate board on the evolution of the radial wall jet, the radial distributions of r location of the half-height for different nozzle height-to-plate distance ratios (H/D) are presented in Figure 19.

The impingement region gradually decreased with an increase in nozzle height H . In general, the half-height value increases with the increase in nozzle height H , which is in good agreement with the results obtained by Knowles and Myszko [14] and Cooper et al. [13]. The current results are similar to those of Copper et al. [13] in the impingement region and are in good agreement with the data obtained by Knowles and Myszko [14] in the radial wall jet region. However, when the outflow height $H/D < 2$ or $3 < H/D < 5$, the half-height did not change significantly. Therefore, for the range in which these nozzle heights are located, the outflow height has little effect on the half-height value.

The effect of the nozzle height on the radial evolution of the maximum velocity is shown in Figure 20. It can be observed that with an increase in the nozzle height, the closer the radial position of the maximum wind speed to the stagnation point, the lower the maximum wind speed. Compared with existing literature data, the current simulation results are similar to the data reported by Knowles and Myszko [14] in the impingement region, and they agree well with the results obtained by Cooper et al. [13] for $r > 1.5D$. The influence of the nozzle height on the maximum velocity decay also exhibited step characteristics, similar to the influence on the half-height.

Based on the data collected during the JAWS project, Hjelmfelt [46] found that the average ratio of the downburst outflow height to diameter is approximately 1.5. It can be observed from the above results that the nozzle height has no significant effect on the jet spread and velocity decay of the radial wall jet near this average ratio. Therefore, when the

FIGURE 19: Radial growth of jet half-height for different nozzle heights.

FIGURE 20: Decay of maximum mean radial velocity for different nozzle heights.

radial wall jet method is used to study the downburst, the influence of the outflow height can be ignored. The most widely used nozzle height is $2D$ [35, 56].

4. Validation of the 2D Assumption for the Downburst Outflow

The outflow of the stationary ideal downburst radially spreads outward from the stationary point. When the radial wall jet and plane wall jet are used to simulate the outflow of the downburst, although similar wind profiles can be generated, it can be seen from the above studies that there are differences between the jet spreads and velocity decays. In comparison, radial wall jets have faster decay rates. To evaluate the accuracy of approximating a 3D downburst outflow with a 2D wall jet, the configuration of the transmission tower and downburst, as presented in Figure 21, was examined. A single-span transmission tower-line system with a span length of S was used for the analysis. The distance r_A and angle θ were used to define the location of the center of the tower relative to the stagnation of the downburst.

The differences between the half-heights and maximum velocities of locations A and B are defined as $\Delta y_{1/2}$ and ΔU_m , respectively. The deviation of the half-height differences of the downburst outflow obtained using the two methods is

$$\gamma_1 = \Delta y_{(1/2)\text{-radial}} - \Delta y_{(1/2)\text{-plane}}, \quad (9)$$

and the deviation of the maximum velocity differences of the downburst outflow obtained using the two methods is

$$\gamma_2 = \Delta U_{m\text{-radial}} - \Delta U_{m\text{-plane}}. \quad (10)$$

A downburst case was assumed with $D = 1000$ m and $U_j = 80$ m/s to investigate the difference between the impinging jet and plane wall jet. Tower A is located at a distance of $1.8D$ from the stagnation point. Using simple trigonometry, the radial distance of tower B, which is the location of the stagnation point of the downburst relative to the center of the tower, can be evaluated. When the plane wall jet method is used, x_a is equal to r_a . Using simple trigonometry, the value of x_b can be calculated. In addition, the height of the jet nozzle (b) can be estimated according to the height of the maximum wind velocity (y_m) and the diameter of the downburst (D). The detailed algorithm is provided in Appendix B. Therefore, the length scale deviations between the radial and plane wall jet methods can be evaluated using equations (5), (7), and (9). The velocity scale deviations can be evaluated using equations (6), (8), and (10). For various combinations of angles (θ) and span lengths (S), the ratios of the length scale deviations to the half-height at location A for the radial wall jet are listed in Table 4, and the ratios of the velocity scale deviations to the maximum velocity at location A for the radial wall jet are listed in Table 5.

Assuming a ratio of less than 5% as an acceptable value, all the values in Table 4 and the upper left (unshaded) values in Table 5 indicate that the two-dimensional (2D) assumption is valid for wide structures. For example, for a 200 m transmission tower-line system, there are no clear

FIGURE 21: Horizontal projection of transmission tower and downburst.

TABLE 4: γ_1 as a percentage of half-height of radial wall jet at location A.

	S (m)				
θ	100	200	300	400	500
0°	0.0	0.2	0.5	0.8	1.3
15°	0.5	0.8	1.0	1.0	0.7
30°	1.0	1.7	2.1	2.2	2.1
45°	1.4	2.3	2.9	3.1	3.0
60°	1.7	2.8	3.5	3.7	3.6
75°	1.8	3.1	3.8	4.1	3.9
90°	1.9	3.2	3.9	4.2	4.0

TABLE 5: γ_2 as a percentage of the maximum velocity for the radial wall jet at location A.

	S (m)				
θ	100	200	300	400	500
0°	0.1	0.5	1.1	2.0	3.0
15°	0.8	1.9	3.1	4.5	6.1
30°	1.4	3.1	4.9	6.8	8.7
45°	2.0	4.1	6.3	8.5	10.8
60°	2.4	4.8	7.3	9.8	12.2
75°	2.6	5.3	8.0	10.5	13.0
90°	2.7	5.5	8.2	10.8	13.3

differences between the length and velocity scales of the radial and plane wall jets. The simplified 2D approach appears to be effective for simulating downbursts. Using the plane wall jet method, a large-scale wind tunnel test can be performed based on the traditional atmospheric boundary layer wind tunnel.

FIGURE 22: Control volume used to illustrate discretization of a scalar transport equation: (a) p-control volume, (b) u_e -control volume, (c) v_s -control volume.

5. Conclusions

In the present study, turbulent radial and plane wall jets were simulated numerically using RSM. The numerical results were compared with previous experimental measurements in the literature, and the effects of different parameters on the length and velocity scales were systematically evaluated. Based on the CFD results, it is valid to approximate a downburst outflow with a 2D assumption for a transmission line under specific conditions. The main findings of this study are summarized as follows:

- (1) The computed results show that the Reynolds stress model accurately predicts the behaviors of radial and plane wall jets. The predictions from the current simulation agree well with the experimental data available in the literature. Compared with the existing experimental results, the maximum difference was approximately 12%. Both radial and plane wall jet methods can effectively simulate the characteristics of the mean velocity profile of the downburst outflow.
- (2) The decay of the maximum velocity and rate of jet spread for the radial wall jet are independent of the Reynolds number for a fixed nozzle height. The nozzle height has a clear effect on the evolution of the radial wall jet. However, when the value of H/D is approximately less than 2, which includes the average ratio of cloud base height to the diameter of most downbursts, the influence of the nozzle can be ignored.
- (3) The decay of the maximum velocity and the half-height of the plane wall jet are dependent on the Reynolds number below a critical value, $Re_{cr} = 60,000$. Above Re_{cr} , the flow becomes asymptotically independent of the Reynolds number. The influence of the Reynolds number can thus be neglected when the plane wall jet is used to simulate the downburst outflow for $Re > Re_{cr}$. To improve the usability of the plane wall jet approach, the shape functions of scale parameters were proposed.
- (4) Co-flow has a significant influence on the plane wall jet. With an increase in the velocity ratio, the jet spread and decay of the maximum velocity gradually

slow down. When the velocity ratio increases from 0.1 to 0.35, the value of B_p decreases by 52.7%.

- (5) Within the span length of a conventional transmission tower-line system, the discrepancy between the downburst outflows simulated using the plane and radial wall jet approaches in the longitudinal direction can be neglected. It is valid to approximate the downburst outflow with a 2D assumption from the perspective of the longitudinal evolution of the flows. In large-scale facilities, the profile can be optimized by controlling the initial flow field conditions, and the plane wall jet method can produce a flow field several times larger than that of the radial wall jet method.

Appendix

A. The SIMPLEC algorithm

The SIMPLEC algorithm uses the relationship between the velocity and pressure corrections to enforce mass conservation and obtain the pressure field.

To provide a brief review of the SIMPLEC method, the staggered grid shown in Figure 22 is used. The discretized u -momentum equation can be written as follows:

$$a_e u_e = \sum \sum a_{nb} u_{nb} + b_e + A_e (p_P - p_E), \quad (A.1)$$

where p is the pressure, A_e is the area of the face of the P-control volume at e , and a_e is the coefficient of the finite-volume equations. A pressure field p^* is assumed to initiate the SIMPLE calculation process. The u^* velocity is obtained by solving the u -momentum equations and satisfy the following equation:

$$a_e u_e^* = \sum \sum a_{nb} u_{nb}^* + b_e + A_e (p_P^* - p_E^*). \quad (A.2)$$

However, the u^* velocities from equation (A.2) in general do not satisfy the continuity condition. To correct the u^* field, the estimated pressure is corrected by considering $p' = p - p^*$. Subtracting equation (A.2) from equation (A.1) gives

$$a_e u_e' = \sum \sum a_{nb} u_{nb}' + A_e (p_P' - p_E'), \quad (A.3)$$

where p' and u' are the pressure and velocity corrections, respectively. The correct pressure p and velocity u can be written as

$$\begin{aligned} u &= u^* + u', \\ p &= p^* + p'. \end{aligned} \quad (\text{A.4})$$

Subsequently, the u -velocity correction equation of SIMPLEC is given by

$$u_e = d_e (p'_P - p'_E), \quad (\text{A.5})$$

where

$$d_e = \frac{A_e}{a_e - \sum a_{nb}}. \quad (\text{A.6})$$

For the control volume shown in Figure 22, the continuity equation satisfies

$$(\rho u A)_w - (\rho u A)_e + (\rho v A)_s - (\rho u A)_n = 0. \quad (\text{A.7})$$

Substituting the correct u into the continuity equation gives the following result:

$$a_P p'_P = a_E p'_E + a_W p'_W + a_N p'_N + a_S p'_S + b, \quad (\text{A.8})$$

where

$$\begin{aligned} a_E &= (\rho A d)_e & a_W &= (\rho A d)_w, \\ a_N &= (\rho A d)_n & a_S &= (\rho A d)_s, \\ a_P &= a_E + a_W + a_N + a_S, \\ b &= (\rho u^* A)_w - (\rho u^* A)_e + (\rho v^* A)_s - (\rho u^* A)_n. \end{aligned} \quad (\text{A.9})$$

The pressure correction p' can be obtained using equation (A8). Subsequently, the correct velocity field can be obtained.

B. Evaluation of Effective Downdraft Diameter for the Plane Wall Jet

Previous studies have shown that the maximum mean velocity occurs at a height of less than $0.05D$ [28, 45, 58]. For the radial wall jet, the value of y_m obtained from the current RSM results at a distance of $1.5D$ from the stagnation point ($x = 1.5D$) is $0.03D$. The influences of Reynolds numbers on the maximum velocity and location are negligible when $Re > 60,000$. An effective plane wall jet downdraft diameter can be determined based on the height of the maximum velocity, as expressed in equation (B1):

$$\begin{aligned} D_{\text{equ}} &= \frac{y_m}{0.03} \\ &= \frac{(0.0133x + 0.131b)}{0.03}. \end{aligned} \quad (\text{B.1})$$

For example, the nozzle height of the plane wall jet is 0.03 m in the current simulation; thus, the effective plane wall jet downdraft diameter is evaluated as 0.53 m at the downstream distance of $x = 30b$.

Data Availability

The data used to support the findings of this study are included within the article.

Conflicts of Interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

Acknowledgments

The work described in this paper was supported by National Natural Science Foundation of China (Grant nos. 52008070 and 51778097), by the Research Foundation of Chongqing University of Science and Technology (Grant no. ckrc2019039), and by the Science and Technology Research Program of Chongqing Municipal Education Commission (Grant no. KJQN202001511).

References

- [1] B. E. Launder and W. Rodi, "The turbulent wall jet measurements and modeling," *Annual Review of Fluid Mechanics*, vol. 15, no. 1, pp. 429–459, 1983.
- [2] S. A. Bagherzadeh, E. Jalali, M. M. Sarafraz et al., "Effects of magnetic field on micro cross jet injection of dispersed nanoparticles in a microchannel," *International Journal of Numerical Methods for Heat and Fluid Flow*, vol. 30, no. 5, pp. 2683–2704, 2019.
- [3] E. Jalali, O. A. Akbari, M. M. Sarafraz, T. Abbas, and M. R. Safaei, "Heat transfer of oil/MWCNT nanofluid jet injection inside a rectangular microchannel," *Symmetry*, vol. 11, no. 6, p. 757, 2019.
- [4] E. Savory, G. A. R. Parke, M. Zeinoddini, N. Toy, and P. Disney, "Modelling of tornado and microburst-induced wind loading and failure of a lattice transmission tower," *Engineering Structures*, vol. 23, no. 4, pp. 365–375, 2001.
- [5] A. Y. Shehata, A. A. E. Damatty, and E. Savory, "Finite element modeling of transmission line under downburst wind loading," *Finite Elements in Analysis and Design*, vol. 42, no. 1, pp. 71–89, 2005.
- [6] P. Bradshaw, "The normal impingement of a circular air jet on a flat surface," *Wear*, vol. 5, no. 1, 1962.
- [7] G. S. Wood, K. C. S. Kwok, N. A. Motteram, and D. F. Fletcher, "Physical and numerical modelling of thunderstorm downbursts," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 89, no. 6, pp. 535–552, 2001.
- [8] W. E. Lin and E. Savory, "Large-scale quasi-steady modelling of a downburst outflow using a slot jet," *Wind and Structures*, vol. 9, no. 6, pp. 419–440, 2006.
- [9] W. E. Lin and E. Savory, "Physical modelling of a downdraft outflow with a slot jet," *Wind and Structures An International Journal*, vol. 13, no. 5, pp. 385–412, 2010.
- [10] B. E. Launder and W. Rodi, "The turbulent wall jet," *Progress in Aerospace Sciences*, vol. 19, pp. 81–128, 1981.
- [11] I. Z. Naqavi, J. C. Tyacke, and P. G. Tucker, "Direct numerical simulation of a wall jet: flow physics," *Journal of Fluid Mechanics*, vol. 852, pp. 507–542, 2018.
- [12] R. van Hout, V. Rinsky, and Y. G. Grobman, "Experimental study of a round jet impinging on a flat surface: flow field and vortex characteristics in the wall jet," *International Journal of Heat and Fluid Flow*, vol. 70, pp. 41–58, 2018.
- [13] D. Cooper, D. C. Jackson, B. E. Launder, and G. X. Liao, "Impinging jet studies for turbulence model assessment-I. Flow-field experiments," *International Journal of Heat and Mass Transfer*, vol. 36, no. 10, pp. 2675–2684, 1993.

- [14] K. Knowles and M. Myszkowski, "Turbulence measurements in radial wall-jets," *Experimental Thermal and Fluid Science*, vol. 17, no. 1–2, pp. 71–78, 1998.
- [15] S. Yao, Y. Guo, N. Jiang, and J. Liu, "An experimental study of a turbulent jet impinging on a flat surface," *International Journal of Heat and Mass Transfer*, vol. 83, pp. 820–832, 2015.
- [16] H. Abrahamsson, B. Johansson, and L. Löfdahl, "A turbulent plane 2-dimensional wall-jet in a quiescent surrounding," *European Journal of Mechanics - B: Fluids*, vol. 13, no. 5, pp. 533–556, 1994.
- [17] J. G. Eriksson, R. I. Karlsson, and J. Persson, "An experimental study of a two-dimensional plane turbulent wall jet," *Experiments in Fluids*, vol. 25, no. 1, pp. 50–60, 1998.
- [18] N. Rostamy, D. J. Bergstrom, D. Sumner, and J. D. Bugg, "The effect of surface roughness on the turbulence structure of a plane wall jet," *Physics of Fluids*, vol. 23, no. 8, Article ID 085103, 2011.
- [19] I. Wygnanski, Y. Katz, and E. Horev, "On the applicability of various scaling laws to the turbulent wall jet," *Journal of Fluid Mechanics*, vol. 234, no. 1, pp. 669–690, 1992.
- [20] M. D. Zhou and I. Wygnanski, "Parameters governing the turbulent wall jet in an external stream," *AIAA Journal*, vol. 31, no. 5, pp. 848–853, 1993.
- [21] T. Tanaka and E. Tanaka, "Experimental studies of a radial turbulent jet: 2nd report, wall jet on a flat smooth plate," *Bulletin of Jsme*, vol. 20, no. 140, pp. 209–215, 1977.
- [22] R. Banyassady and U. Piomelli, "Interaction of inner and outer layers in plane and radial wall jets," *Journal of Turbulence*, vol. 16, no. 5, pp. 460–483, 2015.
- [23] T. Guo, M. J. Rau, P. P. Vlachos, and S. V. Garimella, "Axisymmetric wall jet development in confined jet impingement," *Physics of Fluids*, vol. 29, no. 2, Article ID 025102, 2017.
- [24] P. Fillingham and I. V. Novoselov, "Wall jet similarity of impinging planar underexpanded jets," *International Journal of Heat and Fluid Flow*, vol. 81, Article ID 108516, 2020.
- [25] S. Pieris, X. Zhang, S. Yarusevych, and S. D. Peterson, "Vortex dynamics in a normally impinging planar jet," *Experiments in Fluids*, vol. 60, no. 5, 2019.
- [26] Z. Tang, N. Rostamy, D. J. Bergstrom, J. D. Bugg, and D. Sumner, "Incomplete similarity of a plane turbulent wall jet on smooth and transitionally rough surfaces," *Journal of Turbulence*, vol. 16, no. 11, pp. 1076–1090, 2015.
- [27] M. Khashehchi and Z. Harun, "Accuracy of tomographic particle image velocimetry data on a turbulent round jet," *International Journal of Heat and Fluid Flow*, vol. 77, pp. 61–72, 2019.
- [28] M. T. Chay and C. W. Letchford, "Pressure distributions on a cube in a simulated thunderstorm downburst-Part A: stationary downburst observations," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 90, no. 7, pp. 711–732, 2002.
- [29] C. Junayed, C. Jubayer, D. Parvu, D. Romanic, and H. Hangan, "Flow field dynamics of large-scale experimentally produced downburst flows," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 188, pp. 61–79, 2019.
- [30] A. C. McConville, M. Sterling, and C. J. Baker, "The physical simulation of thunderstorm downbursts using an impinging jet," *Wind and Structures An International Journal*, vol. 12, no. 2, pp. 133–149, 2009.
- [31] A. Dejoan and M. A. Leschziner, "Large eddy simulation of a plane turbulent wall jet," *Physics of Fluids*, vol. 17, no. 2, Article ID 025102, 2005.
- [32] W. E. Lin, E. Savory, R. P. McIntyre, C. S. Vandelaar, and J. P. C. King, "The response of an overhead electrical power transmission line to two types of wind forcing," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 100, no. 1, pp. 58–69, 2012.
- [33] W. E. Lin, L. G. Orf, E. Savory, and C. Novacco, "Proposed large-scale modelling of the transient features of a downburst outflow," *Wind and Structures*, vol. 10, no. 4, pp. 315–346, 2007.
- [34] Z. Yan, Y. Zhong, X. Cheng, R. P. McIntyre, and E. Savory, "A numerical study of a confined turbulent wall jet with an external stream," *Wind and Structures*, vol. 27, no. 2, pp. 101–109, 2018.
- [35] A. Sengupta and P. P. Sarkar, "Experimental measurement and numerical simulation of an impinging jet with application to thunderstorm microburst winds," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 96, no. 3, pp. 345–365, 2008.
- [36] D. C. Wilcox, *Turbulence Modeling for CFD*, Dcw Industries, Incorporated, CA, USA, Second edition, 1998.
- [37] D. C. Wilcox, *Turbulence Modeling for CFD*, Dcw Industries, CA, USA, Third edition, 2006.
- [38] H. Versteeg and W. Malalasekera, *An Introduction to Computational Fluid Dynamics*, Prentice-Hall, NJ, USA, 2007.
- [39] M. Safaei, M. Goodarzi, and M. Mohammadi, "Numerical modeling of turbulence mixed convection heat transfer in air filled enclosures by finite volume method," *The International Journal of Multiphysics*, vol. 5, no. 4, pp. 307–324, 2011.
- [40] J. P. Van Doormaal and G. D. Raithby, "Enhancements of the simple method for PREDICTING incompressible fluid flows," *Numerical Heat Transfer*, vol. 7, no. 2, pp. 147–163, 1984.
- [41] Z. Yan, Y. Zhong, W. E. Lin, E. Savory, and Y. You, "Evaluation of RANS and LES turbulence models for simulating a steady 2-D plane wall jet," *Engineering Computations*, vol. 35, no. 1, pp. 211–234, 2018.
- [42] W. K. George, H. Abrahamsson, J. Eriksson, R. I. Karlsson, L. Löfdahl, and M. Wosnik, "A similarity theory for the turbulent plane wall jet without external stream," *Journal of Fluid Mechanics*, vol. 425, pp. 367–411, 2000.
- [43] M. T. Chay, F. Albermani, and R. Wilson, "Numerical and analytical simulation of downburst wind loads," *Engineering Structures*, vol. 28, no. 2, pp. 240–254, 2006.
- [44] D. D. Vicroy, "Assessment of microburst models for down-draft estimation," *Journal of Aircraft*, vol. 29, no. 6, pp. 1043–1048, 1992.
- [45] Z. Xu and H. Hangan, "Scale, boundary and inlet condition effects on impinging jets," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 96, no. 12, pp. 2383–2402, 2008.
- [46] M. R. Hjelmfelt, "Structure and life cycle of microburst outflows observed in Colorado," *Journal of Applied Meteorology*, vol. 27, no. 8, pp. 900–927, 1988.
- [47] R. P. McIntyre, *The Effect Of Inlet Geometry On the Development Of a Plane Wall Jet*, PhD Thesis, School of Graduate and Postdoctoral Studies, University of Western Ontario, London Canada, 2011.
- [48] G. Xu and R. Antonia, "Effect of different initial conditions on a turbulent round free jet," *Experiments in Fluids*, vol. 33, no. 5, pp. 677–683, 2002.
- [49] R. M. Oseguera and R. L. Bowles, *A Simple, Analytic 3-dimensional Downburst Model Based on Boundary Layer Stagnation Flow* (NASA Technical Memorandum 100632, National Aeronautics and Space Administration, Washington D. C., USA, 1988.
- [50] W. H. Schwarz and W. P. Cosart, "The two-dimensional turbulent wall-jet," *Journal of Fluid Mechanics*, vol. 10, no. 4, 1961.

- [51] S. H. A. Ben, S. Habli, N. S. Mahjoub, P. Bournot, and G. L. Palec, "A numerical study of a plane turbulent wall jet in a coflow stream," *Journal of Hydro-Environment Research*, vol. 12, pp. 16–30, 2016.
- [52] M. E. Schneider and R. J. Goldstein, "Laser Doppler measurement of turbulence parameters in a two-dimensional plane wall jet," *Physics of Fluids*, vol. 6, no. 9, pp. 3116–3129, 1994.
- [53] W. E. Lin, *Validation Of a Novel Downdraff Outflow Simulator: A Slot Jet Wind Tunnel [Doctor of Philosophy]*, The University of Western Ontario, Canada, USA, 2010.
- [54] G. I. Barenblatt, A. J. Chorin, and V. M. Prostokishin, "The turbulent wall jet: a triple-layered structure and incomplete similarity," *Proceedings of the National Academy of Sciences*, vol. 102, no. 25, pp. 8850–8853, 2005.
- [55] R. McIntyre, E. Savory, H. Wu, and D. S.-K. Ting, "The effect of the nozzle top lip thickness on a two-dimensional wall jet," *Journal of Fluids Engineering*, vol. 141, no. 5, Article ID 051106, 2019.
- [56] C. Li, Q. S. Li, Y. Q. Xiao, and J. P. Ou, "A revised empirical model and CFD simulations for 3D axisymmetric steady-state flows of downbursts and impinging jets," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 102, pp. 48–60, 2012.
- [57] M. J. Tummers, J. Jacobse, and S. G. J. Voorbrood, "Turbulent flow in the near field of a round impinging jet," *International Journal of Heat and Mass Transfer*, vol. 54, no. 23–24, pp. 4939–4948, 2011.
- [58] J. Kim and H. Hangan, "Numerical simulations of impinging jets with application to downbursts," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 95, no. 4, pp. 279–298, 2007.