



Article Numerical Evaluation of the Flow Field of an Isothermal Dual-Corner Impinging Jet for Building Ventilation

Arman Ameen ^{1,*}, Haruna Yamasawa ² and Tomohiro Kobayashi ³

- ¹ Faculty of Engineering and Sustainable Development, Department of Building Engineering, Energy Systems and Sustainability Science, University of Gävle, 801 76 Gävle, Sweden
- ² Faculty of Engineering Sciences, Department of Advanced Environmental Science and Engineering, Kyushu University, 6-1 Kasuga-koen, Kasuga 816-8580, Japan
- ³ Department of Architectural Engineering, Division of Global Architecture, Osaka University, 2-1 Yamadaoka, Suita 565-0871, Japan
- * Correspondence: arman.ameen@hig.se

Abstract: The corner impinging jet ventilation is a new air distribution system for use in office environments. This study reports the mean flow field behavior of dual isothermal corner-placed inlets based on an impinging jet in a square-shaped room with the size of 7.2 m × 7.2 m. A detailed numerical study is carried out to evaluate the influence the different configuration parameters, such as the inlet placement, same side or opposite side, and supply airflow rate, have on the flow field. The results show that the highest velocity peak for all cases is obtained at *x* = 0.5 m and the lowest at *x* = 3.5 m. The velocity profiles development remains similar when increasing the flow rate. For the zone evaluation, the results show that Case 1 and 2 (V = 20 L/s) meet the requirement of not exceeding 0.15 m/s during the heating season in the occupied zone according the BBR standard both for same-side and opposite-side configurations. For Case 4, the optimal placement of the inlets is opposite to each other when V = 30 L/s for the BBR requirements. Case 1, 2, 3, 4, 5, and 7 all meet the requirement of not exceeding 0.25 m/s during the cooling season both for the same-side and opposite-side configurations. For Case 8, the optimal placement of the inlets is opposite to each other when V = 50 L/s.

Keywords: dual-corner impinging jet ventilation; computational fluid dynamic; inlet configuration

1. Introduction

One of the most important aspects of buildings are to provide the occupants with adequate thermal comfort and a healthy environment [1]. This is done primarily with heating, ventilation, and air conditioning (HVAC) systems. There are many types of ventilation systems, and each system can utilize a variety of air distribution systems (ADS). The most common type is called the mixing ventilation (MV) [2–5], which is also the most common system installed around the world. In the last two decades, new ADS systems have been developed, such as displacement ventilation (DV) [4–7] and impinging jet ventilation (IJV) [4,5,7-10]. These systems are classified as stratified ventilation. This is due to the temperature stratification these ADSs create, especially during the cooling season. DV usually works with low-to-medium velocities when air enters the room at floor height. In the heating season, this system is very restricted due to the buoyancy effect. To overcome this shortcoming, an IJV was introduced which combines the advantages of both the MV and DV systems. In an IJV, air with high momentum is discharged downwards, distributing fresh air over the floor area farther into the room compared to DV. This suggests that IJV can be used for both cooling and heating in larger spaces than can DV systems. Yamasawa et al. [8] compared the center- and corner-placed inlet IJVs and evaluated these setups based on several key indexes. The space configuration for this study was based on an office and occupants number ranging from 9 to 36. When evaluating terminal



Citation: Ameen, A.; Yamasawa, H.; Kobayashi, T. Numerical Evaluation of the Flow Field of an Isothermal Dual-Corner Impinging Jet for Building Ventilation. *Buildings* 2022, *12*, 1767. https://doi.org/10.3390/ buildings12101767

Academic Editor: Zohir Younsi

Received: 17 August 2022 Accepted: 19 October 2022 Published: 21 October 2022

Publisher's Note: MDPI stays neutral with regard to jurisdictional claims in published maps and institutional affiliations.



Copyright: © 2022 by the authors. Licensee MDPI, Basel, Switzerland. This article is an open access article distributed under the terms and conditions of the Creative Commons Attribution (CC BY) license (https:// creativecommons.org/licenses/by/ 4.0/). configurations placement, either at the corners or at the wall center, the results show a small difference. In a recent paper, Staveckis and Borodinecs [11] evaluated an office room equipped with an IJV. They tested this ADS both under summer and winter climate conditions. They also included changes in human positions in the room and tested different supply duct geometries. Their study showed that the shape of the duct had little effect on the contaminant and heat-removal effectiveness. The study also showed that the optimal placement for the occupant depended on the supply temperature and flow velocity.

In 2019, Ameen et al. [2,3] made additional modifications to the standard IJV by placing the supply inlet in the corners of the room. In these studies, triangle-shaped inlets were used in order to simplify both the installation and material used for the inlet itself. This was also done to facilitate more space to use in the room (for furniture, desks, etc.) as well the possibility to place the inlets at walls that had windows installed. The authors named this configuration corner impinging jet ventilation (CIJV). The researchers evaluated and compared this system against two other systems, DV and MV. The findings of the study showed that the CIJV air distribution system operated very similarly to a DV system and performed slightly better when examining the draft rate. Recently, Ameen et al. [12] numerically examined a CIJV with a single inlet configuration. The aim of the study was to examine the air flow field of an isothermal CIJV in an empty office space by using an experimentally validated CFD model. Several turbulence models were tested and an RNG $k - \varepsilon$ was chosen. Several parameters and indexes were evaluated, such as velocity profiles and velocity decay, as well as the spreading rate along the diagonal centerline of the computational space. Additionally, a large parametric study was run that included various inlet shapes, different inlet flow rates, different inlet areas, and multiple inlet discharge heights above the floor level. Moreover, a second evaluation was carried out to evaluate the maximum velocity reach at different distances diagonal from the inlet by only using the triangle-shaped-based inlet. In the final part of the study, regression analyses were done to create equations to predict the jet spreading rate and maximum velocity decay, and also to show the connection between the design parameters and how these affect the flow field in general. However, this study only focused on one inlet configuration.

To the best of the authors' knowledge, they have not seen any major studies that examine the detailed isothermal flow behavior of CIJVs in multiple inlet configurations, especially in a confined room setting. The aim of this study is to examine the flow field of a dual isothermal CIJV with two inlet configurations in a room by using a validated CFD model. This evaluation will be conducted for two main configurations: one is a two-inlet configuration where the inlets are placed on the same wall side, and a second configuration where the inlets are placed on the same wall side, and a second configuration where the inlets are placed on the same wall side, and a second configuration where the inlets are placed on the same wall side, and a second configuration where the inlets are placed on the same wall side, and a second configuration where the inlets are placed on the same wall side, and a second configuration where the inlets are placed on the same wall side, and a second configuration where the inlets are placed on the same wall side, and a second configuration where the inlets are placed on the same wall side, and a second configuration where the inlets are placed at the opposite corners of the room. The turbulence model RNG $k - \varepsilon$ was chosen based on its capability of predicting the impinging jet flow in previous studies. A parametric configuration with a different supply flow rate as well as the different placements of the inlets in the space will be done. The flow field in the space will be analyzed by evaluating the air velocity profiles and the velocity contours in the occupied region of the room.

2. Methodology

2.1. Modeling Equations

The CFD simulations were performed based on the following assumptions and limitations: steady-state (all the states of dynamic system have reached an equilibrium state), incompressible air modeling, the air is turbulent, and the model is under isothermal condition. This study is based on these assumptions and limitations. The 3D Reynolds-averaged Navier–Stokes (RANS) equations are as follows:

$$\frac{\partial U_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial(\rho U_j U_i)}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \mu \nabla^2 U_i + \frac{\partial}{\partial x_j} \left(-\rho \overline{u'_i u'_j}\right)$$
(2)

where ρ is the fluid density, U_i is the mean velocity component in x_i direction x_i , P is the pressure, μ is dynamic viscosity, and u' prime is the fluctuating component of velocity.

Equation (2) shows the Reynolds stresses (u'_i, u'_j) that are given by the Boussinesq hypothesis:

$$-\rho \overline{u'_i u'_j} = -2\mu_t S_{ij} + \frac{2}{3} \delta_{ij} \rho k \tag{3}$$

where *k* is the turbulent kinetic energy. Here, $k = 0.5 \cdot \overline{u'_i u'_j}$. The eddy viscosity is μ_t and δ_{ij} is the Kronecker symbol; $\delta_{ij} = 1$ if I = j and $\delta_{ij} = 0$ if $I \neq j$.

The strain rate tensor, S_{ij} , is calculated as:

$$S_{ij} = 0.5 \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right)$$
(4)

2.2. Turbulence Models

The turbulence model for this study was chosen based on a previous study [12] that tested three different types of turbulence models, RNG $k - \varepsilon$, SST $k - \omega$, and $\overline{v^2} - f$, in order to evaluate their viability of predicting the impinging jet flow fields. Hence, the RNG $k - \varepsilon$ was chosen as the turbulence model in this study based on the result of that study, which is also similar to some other studies that have evaluated the impinging jet settings [13–15]. For a more in-depth mathematical description of the RNG $k - \varepsilon$ turbulence model, see [14,15].

2.3. Numerical Setup

The software Ansys Icepac 2022 R2 was used in order to create the space geometry and generate the appropriate mesh structure. Ansys Fluent 2022 R2 was used for the numerical simulation of the cases. SIMPLE was used as the pressure–velocity coupling scheme. The least squares cell-based method was used to solve the gradients in the model. The pressure term used was the second-order upwind discretization scheme. The under-relaxation factor of 0.3 was used at the beginning of the simulation and that was increased 0.7 at the end-stage. As for the momentum term, the second-order upwind discretization scheme was used with an under-relaxation factor of 0.7 at the beginning of the simulation. The under-relaxation factor of 1 the second-order upwind discretization scheme was used both for the turbulent kinetic energy and the specific dissipation rate. The convergence criteria used for the continuity was less than 10^{-3} , and for the other terms, it was less than 10^{-4} .

2.4. Mesh Configuration and Mesh Independence Test

A nonuniform grid distribution was used in this study, with the refinement for the mesh being focused at and around the inlets, side walls, and floor. Three different mesh densities were examined in order to optimize the result accuracy as well as reduce the required computational load for the simulations. The total number of structured hexahedral cells contained within the three tested models were 10.43, 14.60, and 20.44 million. The turbulence model used for the mesh independency test was RNG $k - \varepsilon$. The near-wall model used was the enhanced wall treatment. The difference in the results between the different cell densities was determined by evaluating the velocity at various points in the domain. The root-mean-square error (RMSE) was calculated by using Equation (5). In total, 52 measurement points were chosen at distances 0.5, 1.0, 1.5, and 2.0 m. At each distance, 13 measuring positions were measured. In Equation (5), *n* is the total number of selected points and U_{in} is the nominal velocity of the inlet, which is set to 1.5 m/s (20 L/s for each

inlet). $U_{i,BG}$ is the result at lower cell count and $U_{i,RG}$ is the numerical result at higher cell count.

$$RMSE = \frac{\sum_{i=1}^{n} \sqrt{\left(\frac{U_{i,BG} - U_{i,RG}}{U_{in}}\right)^2}}{n} \cdot 100$$
(5)

The result of the different mesh densities showed a small gain between the last two models (14.60 and 20.44 million). The RMSE between 10.43 and 14.60 million was 2.83%, but only 0.89% between 14.60 and 20.44 million; therefore, the model with 14.60 million cells was chosen for this study. Figure 1a shows a top sideview of the room mesh. Figure 1b shows the y^+ distribution, which is resulted to be $y^+ \leq 1$ in the whole room, and Figure 1c shows a closeup of the inlet.



Figure 1. (a) Shows a perspective view over the room mesh where the two inlets are marked with blue and the outlet in the center is market with red. (b) Shows the y^+ distribution in the model. (c) Shows a more detailed view of the corner place inlet.

2.5. CFD Validation

The base model used in this study is from a previous study and was validated by Ameen et al. [12]. An experimental comparison was made between the CFD model and an experimental setup, which compared the air velocity profiles measured at different locations along the centerline of the floor surface, specifically at distances of 0.225, 0.3, 0.5, 0.7, and 1 m from the inlet. Moreover, the jet maximum velocity decay along the centerline of the floor was compared. Several turbulence models were evaluated, RNG $k - \varepsilon$, SST $k - \omega$, and $\overline{v^2} - f$. The results of the validation showed that the predicted jet profiles were in good agreement with the experiential results at each measuring point. The RMSE value was 2.73 % for RNG $k - \varepsilon$, 3.68 % for SST $k - \omega$, and 2.33 % for the $\overline{v^2} - f$ turbulence model. The predicted results of the three turbulence models were very similar to each other. However, some of the models showed better accuracy close to the inlet, while some showed better when measuring farther away from the supply inlet. The validation study concluded that RNG $k - \varepsilon$ was suitable for the study.

2.6. Case Studies

In total, ten simulation cases were chosen, and they were run in order to evaluate the air flow pattern in the room. Detailed case setting and parameters are shown in Table 1. Figure 2 shows all the measurement positions for the velocity profiles in the CFD model and the three different inlet positions. It also shows the different zones of interest (Zones 1–9) with regard to evaluating the flow field. The velocity evaluations are made at the diagonal center from each inlet at the distance of 0.5, 1.0, 1.5, 2.0, 2.5, 3.0, and 3.5 m. Zones 1–9 are within the occupied zone, which starts at a distance of 0.6 m from the wall. Each zone is a square with the size of $2 \text{ m} \times 2 \text{ m}$. When evaluating the same-side configuration, only inlets 1 and 2 are active. When evaluating the opposite-side configuration only inlets 1 and 3 are active. The outlet in the model is placed directly at the center of the model's ceiling. When evaluating the velocity profiles at the diagonal distance from the inlets in the results section, I-1 (Inlet-1 side), I-2 (Inlet-2 side), and I-3 (Inlet-3 side) will be used to indicated which inlet side is evaluated. The discharge height was set to be 0.8 m above the floor level for all of the inlets. The base inlet area was set 0.0133 m², which is the same as in these previous studies [4,5,12,14]. The flow rate was chosen to cover a wide range of flow rates required for this room size, depending on how many occupants are present and what their activity levels are.

Table 1. Case conditions and parameter settings all cases.

Case	Inlet Placement	Total Supply Flow Rate V (L/s)	Supply Flow Rate Per Inlet (L/s)	Inlet U _{in} (m/s)	Re
Case-1	Same Side	20	10	0.75	5705
Case-2	Opposite Side	20	10	0.75	5705
Case-3	Same Side	30	15	1.13	8596
Case-4	Opposite Side	30	15	1.13	8596
Case-5	Same Side	40	20	1.50	11,411
Case-6	Opposite Side	40	20	1.50	11,411
Case-7	Same Side	50	25	1.88	14,302
Case-8	Opposite Side	50	25	1.88	14,302
Case-9	Same Side	60	30	2.26	17,192
Case-10	Opposite Side	60	30	2.26	17,192



Figure 2. The layout of the measurement positions and their distance to each corner part of the room. The figure also shows regions of interest in the form of zones 1–9. On the left side, a zoomed-in picture of the inlet is shown with its dimensions.

The total supply flow rates chosen were 20, 30, 40, 50, and 60 L/s. However, since the room is equipped with two inlets, these values are translated to 10, 15, 20, 25, and 30 L/s per inlet. Every configuration is tested both with the inlets being on the same wall side and on the opposite side, as shown in Figure 3. The outlet of the model is placed at the center of the ceiling. The area of interest is divided into nine zones. Each of these zones will be evaluated in terms of average velocity and maximum velocity at the height of 0.1 m above the floor level.



Figure 3. Shows a perspective view of the two inlet configurations tested. The left side shows the same side evaluation, and the right side shows the opposite side evaluation.

3. Results and Discussion

3.1. Velocity Profile

Figure 4 shows the results of the air velocity development along the centerline at different distances from the corner when evaluating the cases at V = 20 L/s (10 L/s per inlet). The results are presented in dimensionless form, where y/h is the height from the floor divided by the room height (h = 2.67 m) and U/U_{in} is the velocity (U) divided by the nominal inlet velocity (U_{in}). When evaluating the cases at V = 20 L/s, it can be seen that, at x = 0.5 m (Figure 4a), when there is inlet close to the evaluation points, i.e., Case-1 I-1, Case-2 I-1, Case-1 I-2, and Case-2 I-3, the velocity profiles have a similar peak velocity; however, there is a small difference. The peak velocity is 0.53 m/s for Case-1 I-1, 0.54 m/s for Case-2 I-3. The results also show that the jet at x = 0.5 m is mainly contained below y/h = 0.04 for these cases. When comparing Case-1 vs. Case-2, the results show a slightly lower velocity for Case-1. When evaluating the diagonal centerline when there is no inlet in close proximity, the results show that the general velocity in these areas is much lower.

Moreover, when comparing Case-2 I-2 vs. Case-1 I-3, where there is no inlet close to the evaluation points, the results show a slightly higher velocity starting around y/h = 0.01 and upwards for Case-1 I-3 when $x \le 1.0$ m. This indicates that having the inlets on the same side increases the velocity at the region far from the inlet rather than having the inlet on the opposite side at x = 0.5 m.

When evaluating the cases with V = 20 L/s at x = 1.0 m (Figure 4b), the results show that the peak velocity has decreased considerably compared to that at x = 0.5 m. The peak velocity is 0.28 m/s for Case-1 I-1, 0.27 m/s for Case-2 I-1, 0.28 m/s for Case-1 I-2, and 0.27 m/s for Case-2 I-3. It is also worth mentioning that x = 1.0 m is within the occupied zone (zone 1, 7, and 9). The results also show that the jet is now spreading higher above the floor, as the jet width is now reaching y/h = 0.06. Moreover, the difference between Case-2 I-2 and Case-1 I-3 has decreased in terms of overall velocity.

When evaluating the cases with V = 20 L/s at x = 1.5 m (Figure 4c), the results show that the peak velocity has decreased once again as the jet travels farther into the room. The peak velocity is 0.18 m/s for Case-1 I-1, 0.18 m/s for Case-2 I-1, 0.18 m/s for Case-1 I-2, and 0.18 m/s for Case-2 I-3. These results suggests that the differences between the cases almost to disappear when evaluating the velocity at this distance. There is now no noticeable difference between Case-2 I-2 and Case-1 I-3.



Figure 4. (**a**–**g**) Show the velocity changes along the centerline of the floor for all evaluated cases with V = 20 L/s at *x* distances 0.5–3.5 m.

When evaluating the cases with V = 20 L/s at x = 2.0 m (Figure 4d), the results show that the peak velocity has decreased once again as the jet travels even farther into the room. The peak velocity is 0.13 m/s for Case-1 I-1, 0.13 m/s for Case-2 I-1, 0.13 m/s for Case-1 I-2, and 0.13 m/s for Case-2 I-3. The difference between the cases can hardly be seen at this distance. The velocity difference between Case-2 I-2 and Case-1 I-3 at this distance is now reversed compared to those at the points closer to the inlet. At x = 0.5 m, Case-1 I-3 had a slightly higher velocity compared to Case-2 I-2; however, at x = 2.0 m, Case-2 I-2 has a slightly higher velocity compared to Case-1 I-3.

Farther into the room, at x = 2.0-3.5 m, the results show a similar tendency of a reduction in the peak velocity as the jets get farther away from the inlets when there is inlet close to the evaluation points, i.e., Case-1 I-1, Case-2 I-1, Case-1 I-2, and Case-2 I-3. However, when comparing Case-2 I-2 vs. Case-1 I-3, the general velocity of Case-2 I-2 is slightly higher at these distances compared to those at x = 0.5-1.0 m.

Figure 5 shows the results of the air velocity development along the centerline at various distances when evaluating all the cases at V = 30 L/s (15 L/s per inlet). The peak velocities at x = 0.5 m (Figure 5a) are 0.83 m/s for Case-3 I-1, 0.84 m/s for Case-4 I-1, 0.83 m/s for Case-3 I-2, and 0.84 m/s for Case-4 I-3, and the jet is mainly contained below y/h = 0.04 for these cases. When comparing Case-3 (inlet on the same side) vs. Case-4 (inlet on the opposite side) the results show a slightly lower velocity for Case-3. Moreover, when comparing Case-4 I-2 vs. Case-3 I-3, the results show a slightly higher velocity starting around y/h = 0.002 and upwards for Case-3 I-3. When evaluating the cases with V = 30 L/s at x = 1.0 m (Figure 5b), the results show that the peak velocity has decreased considerably compared to that at x = 0.5 m. The max or peak velocity is 0.44 m/s for Case-4 I-3. The results also show that the jet is now spreading higher above the floor, as the jet width is now reaching $y/h \approx 0.06$. Moreover, the difference between Case-4 I-2 and Case-3 I-3 has decreased in terms of overall velocity.



Figure 5. (**a**–**g**) Show the velocity developments along the centerline of the floor for all evaluated cases with V = 30 L/s at *x* distances 0.5–3.5 m.

When evaluating the cases with V = 30 L/s at x = 1.5 m (Figure 5c), the results show that the peak velocity has decreased once again as the jet travels farther into the room. The peak velocity is 0.29 m/s for Case-3 I-1, 0.28 m/s for Case-4 I-1, 0.29 m/s for Case-3 I-2, and 0.28 m/s for Case-4 I-3. These results suggests that there is a small decrease for Case-4 compared to Case-3 at this distance. At this distance, there is now no noticeable difference between Case-3 I-2 and Case-4 I-3.

When evaluating the cases with V = 30 L/s at x = 2.0 m (Figure 5d), the results show that the peak velocity has decreased as the jet travels even farther into the room. The peak velocity is 0.22 m/s for Case-3 I-1, 0.21 m/s for Case-4 I-1, 0.22 m/s for Case-3 I-2, and 0.21 m/s for Case-4 I-3. These results suggests that there is a small decrease for Case-4 compared to Case-3 at this distance. The velocity difference between Case-4 I-2 and Case-3 I-3 at this distance is now reversed compared to those at the points closer to the inlet. At x = 0.5 m Case-3 I-3 had a slightly higher velocity compared to Case-3 I-2; however, at x = 2.0 m, Case-4 I-2 has a slightly higher velocity compared to Case-3 I-3.

Farther into the room, at x = 2.0-3.5 m, the results show similar tendency of a reduction in the peak velocity as the jets gets farther away from the inlets for Case-3 I-1, Case-4 I-1, Case-3 I-2, and Case-4 I-3. However, when comparing Case-4 I-2 vs. Case-3 I-3, the general velocity of Case-4 I-2 is slightly higher at x = 2.0-3.5 m compared to x = 0.5-1.0 m.

Figure 6 shows the results of the velocity development along the centerline at various distances from the corner for cases at V = 40 L/s (20 L/s per inlet). When evaluating the cases at V = 40 L/s, it can be seen that, at x = 0.5 m (Figure 6a), the peak velocity at the points close to the inlet is 1.10 m/s for Case-5 I-1, 1.13 m/s for Case-6 I-1, 1.10 m/s for Case-5 I-2, and 1.14 m/s for Case-6 I-3. The results also show that the jet is mainly contained below y/h = 0.04 for the cases at x = 0.5. When comparing Case-5 vs. Case-6, the results show a slightly lower velocity for Case-5 I-2 when comparing the results at the point with no inlet nearby, i.e., Case-6 I-2 vs. Case-5 I-3, the results show a slightly higher velocity starting around y/h = 0.002 and upwards for Case-5 I-3.



Figure 6. (**a**–**g**) Show the velocity change along the centerline of the floor for all evaluated cases with V = 40 L/s at *x* distances 0.5–3.5 m.

When evaluating the cases with V = 40 L/s at x = 1.0 m (Figure 6b), the results show that the peak velocity has decreased considerably compared to the x = 0.5 m. The peak velocity is 0.59 m/s for Case-5 I-1, 0.59 m/s for Case-6 I-1, 0.59 m/s for Case-5 I-2, and 0.59 m/s for Case-6 I-3. At this distance, the peak velocity is the same for all the cases. The results also show that the jet is now spreading higher above the floor than at x = 0.5 m, as the jet width is now reaching $y/h \approx 0.06$ compared to x = 1.0 m. Moreover, the difference between Case-6 I-2 and Case-5 I-3 has decreased in terms of overall velocity. When evaluating the cases with V = 40 L/s at x = 1.5 m (Figure 6c), the results show the peak velocity is 0.39 m/s for Case-5 I-1, 0.38 m/s for Case-6 I-1, 0.39 m/s for Case-5 I-2, and 0.38 m/s for Case-6 I-3. These results suggests that there is a small decrease for Case-6 compared to Case-5 I-2 and Case-6 I-3.

When evaluating the cases with V = 40 L/s at x = 2.0 m (Figure 6d), the results show the peak velocity is 0.29 m/s for Case-5 I-1, 0.28 m/s for Case-6 I-1, 0.29 m/s for Case-5 I-2, and 0.28 m/s for Case-6 I-3. These results suggests that there is a small decrease for Case-6 compared to Case-5 at this distance. The velocity difference between Case-6 I-2 and Case-5 I-3 at this distance is now reversed compared to those at the points closer to the inlet. At x = 0.5 m, Case-5 I-3 had a slightly higher velocity compared to Case-6 I-2; however, at x = 2.0 m, Case-6 I-2 has a slightly higher velocity compared to Case-5 I-3.

Farther into the room, at x = 2.0-3.5 m, the results show similar tendency of a reduction in the peak velocity as the jets gets farther away from the inlets for Case-5 I-1, Case-6 I-1, Case-5 I-2, and Case-6 I-3. However, when comparing Case-6 I-2 vs. Case-5 I-3, the general velocity of Case-6 I-2 is slightly higher at these distances compared to x = 0.5-1.0 m.

Further evaluation of the velocity profiles for V = 50-60 L/s (Figures 7 and 8) reveals a similar pattern, and as the volume flow is increased, a higher peak velocity is obtained for each *x* distance. These results are also similar to the previous research made for one inlet in



terms of the impinging jet velocity profile development close to the floor area and near the inlet [12].

Figure 7. (**a**–**g**) Show the velocity change along the centerline of the floor for all evaluated cases with V = 50 L/s at *x* distances 0.5–3.5 m.

3.2. Velocity Contour

The velocity contour plots at z = 0.1 m are shown for the cases 1–6 in Figure 9 and 7–10 in Figure 10. In Figure 9a, the result of Case 1 is shown. The inlets on the same side create two distinct regions in front of each inlet. The highest velocity is reached around x = 1.3 m diagonally from the inlet. As the jet was moving farther away from the inlet, the velocity decreased, and when reaching the center area, the two flows and velocities merged (in Zone 2 and 5). Moreover, due to the placement of the inlets, a large portion of the supplied air flow moved along the walls rather than the floor, which the authors believe is due to the Coanda effect. Zones 1 and 3 are fairly unaffected and show very low overall velocities.

In Figure 9b, the result of Case 2 is shown. In this case, the inlets are placed opposite to each other. When compared with Case 1, this configuration forces the jet to spread out more to the sides and upwards in the central occupied zone (Zone 5). In this case, Zones 3 and 7 are fairly unaffected compared to the other regions, even though the air from the center part of the room is pushed into these regions after the two jet stream heads meet each other in the center of the room.

Evaluating Case 3 (Figure 9c), the velocity contour shows a similar profile development as that in Case 1, but with a higher magnitude. The flows that attached to the walls were reaching much farther in this case compared to Case 1. When evaluating Case 4 in Figure 9d, the velocity development is similar to that of Case 2, but with a higher magnitude. Evaluating the other cases, the results show that increasing the flow rate leads to an increase in the velocity, but the overall velocity contour shape at each inlet configuration stays the same regardless of the supply flow rate.





Figure 8. (**a**–**g**) Show the velocity change along the centerline of the floor for all evaluated cases with V = 60 L/s at *x* distances 0.5–3.5 m.

£.,

0.02

3.3. Zone Evaluation

0.4 U/U_{in}

٢

0.02

(a)

0.08

0.06

₹_{0.04}

0.02

In order to quantify the contour plots results, Figures 11 and 12 show the results of the average and maximum velocity for each zone for all cases. In Figure 11, when comparing the results in Zone 9, with the region that has Inlet 1 inside of it, for Case 1 vs. Case 2 (the same inlet flowrate of 10 L/s but different inlet configuration), the results show a slightly lower average velocity for Case 2 (0.07 m/s) than Case 1 (0.08 m/s). When comparing the results in Zone 7–Case 1 vs. Zone 1–Case 2, with the regions that have Inlet 2 or 3 inside them, a similar tendency is shown as those in Zone 9, with 0.08 m/s for Case 1 vs. 0.07 m/s for Case 2. This shows that, in the zone closest to the inlets, the velocity is slightly lower for the opposite-side configurations than for the same-side configuration when inlet flow rate is 10 L/s.

When comparing the results in the zones that are farthest from the inlets, Zones 1 and 3 for Case 1 and Zones 3 and 7 for Case 2, the results show that Zones 3 and 7 for Case 2 have a slightly lower average velocity, 0.020 vs. 0.018 m/s.

When comparing Case 1 vs. Case 2 in Zone 5, the average velocities are 0.04 vs. 0.03 m/s, where the velocity is slightly lower for Case 2 than Case 1. Overall, Case 2 had a lower average velocity in five zones, Zones 3, 5, 7, 8, and 9, compared to Case 1. When evaluating the maximum velocity, the highest velocity for Case 1 is 0.11 m/s in Zones 7 and 9, whereas for Case 2, it is 0.10 m/s in Zones 1 and 9.

Since Zones 9 and 7 of Case 3 and Zones 9 and 2 of Case 4 contain Inlet 1, 2, or 3 inside them, the results in those regions are to be compared to investigate the velocity around the inlet. At the regions around the inlet, the average velocity is 0.12 m/s in Case 1, whereas in Case 2 it is 0.11 m/s. Thus, it is shown that the velocity is slightly lower for the opposite-side configuration (Case 4) than for the same-side configuration (Case 3). On the other hand, the regions that contain the corners without the inlet, i.e., Zones 1 and 3 for Case 3 and Zones 3 and 7 for Case 4, are to be compared to investigate the velocity at the region far from the inlet. The average velocity of the region mentioned above is 0.03 m/s in

Case 3 and 0.02 m/s in Case 4, where the velocity in Case 4 is slightly higher than Case 3. As for the center of the room, i.e., Zone 5, the average velocity in Case 3 is 0.07 m/s and in Case 4 it is 0.06 m/s; thus, the tendency is the same as that in the regions that contain inlets at the corner of the room. Overall, Case 4 had lower average velocity in the four zones, Zones 5, 7, 8, and 9, compared to Case 3. When evaluating the maximum velocity, the results show that for Case 3, Zones 7 and 9 show the highest velocity of 0.16 m/s, and for Case 4, 0.15 m/s in Zones 1 and 9.



Figure 9. (**a**–**f**) Show the velocity contour for Cases 1–6 at z = 0.1 m.



Figure 10. (**a**–**d**) Show the velocity contour for Cases 7–10 at z = 0.1 m.

When comparing Zone 9–Case 5 vs. Zone 9–Case 6 (with the same inlet flow rate of 20 L/s), the results show a slightly lower average velocity for Zone 9–Case 6, at 0.16 vs 0.15 m/s. When comparing Zone 7–Case 5 vs. Zone 1–Case 6, a similar result is shown at 0.16 vs. 0.15 m/s. Similar to the previous evaluation of Case 1 vs. Case 2 and Case 3 vs. Case 4, for the zones closest to the inlets, the velocity is slightly lower for the opposite-side configurations. When comparing the zones that are furthest away from the inlets, Zones 1 and 3 for Case 5 and Zones 3 and 7 for Case 6, the results show that Zones 3 and 7 for Case 6 have a slightly lower average velocity, 0.04 vs. 0.03 m/s. When comparing Case 5 vs. Case 6 in Zone 5, the results show a slightly lower average velocity in five zones, Zones 3, 5, 7, 8, and 9, compared to Case 5. When evaluating the maximum velocity, the results show that for Case 5, Zones 7 and 9 show the highest velocity at 0.21 m/s, and for Case 6, it is 0.20 m/s for Zones 1 and 9.

When comparing Zone 9–Case 7 vs. Zone 9–Case 8 (the same inlet flowrate 25 L/s), the results show a slightly lower average velocity for Zone 9–Case 8, at 0.20 vs. 0.19 m/s. When comparing Zone 7–Case 7 vs. Zone 1–Case 8, a similar result is shown at 0.20 vs. 0.19 m/s. Similar to the previous evaluation of Case 1 vs. Case 2, Case 3 vs. Case 4, and

Case 5 vs. Case 6, for the zones closest to the inlets, the velocity is slightly lower for the opposite-side configurations. When comparing the zones that are farthest from the inlets, Zones 1 and 3 for Case 7 and Zones 3 and 7 for Case 8, the results show that Zones 3 and 7 for Case 8 have a slightly lower average velocity at 0.06 vs. 0.05 m/s. When comparing Case 7 vs. Case 8 in Zone 5, the results show a slightly lower average velocity for Case 8 at 0.12 vs. 0.10 m/s. Overall, Case 8 had a lower average velocity in five zones, Zones 3, 5, 7, 8, and 9, compared to Case 7. When evaluating the maximum velocity, the results show that, for Case 7, Zones 7 and 9 show the highest velocity at 0.26 m/s, and for Case 8, it is 0.25 m/s Zones 1 and 9.



Figure 11. Shows the zone average velocity at z = 0.1 m for all cases.

When comparing Zone 9–Case 9 vs. Zone 9–Case 10 (the same inlet flowrate 30 L/s), the results show a slightly lower average velocity for Zone 9–Case 10 at 0.23 vs 0.22 m/s. When comparing Zone 7–Case 9 vs. Zone 1–Case 10, a similar result is shown at 0.23 vs. 0.22 m/s. Similar to the previous evaluation of the other case comparisons, for the zones closest to the inlets, the velocity is slightly lower for the opposite-side configurations. When comparing the zones that are furthest aways from the inlets, Zones 1 and 3 for Case 9 and Zones 3 and 7 for Case 10, the results show that Zones 3 and 7 for Case 10 have a slightly lower average velocity at 0.06 vs. 0.05 m/s. When comparing Case 9 vs. Case 10 in Zone 5, the results show a slightly lower average velocity for Case 10 at 0.15 vs. 0.13 m/s. Overall, Case 10 had a lower average velocity in five zones, Zones 3, 5, 7, 8, and 9, compared to Case 9. When evaluating the maximum velocity, the results show that, for Case 9, Zones 7 and 9 show the highest velocity at 0.31 m/s, and for Case 10, it is 0.30 m/s for Zones 1 and 9.

When looking at Boverket's building regulation guidelines in Sweden (BBR) [16], the recommended velocities in the occupied zone should not exceed 0.15 m/s during the heating season and 0.25 m/s during the cooling season. When evaluating the various cases based on the BBR guidelines, Cases 1, 2, and 4 meet the requirement of not exceeding 0.15 m/s during heating season. Particularly, Case 4 is of interest, since by placing the inlets opposite to each other, a case with a total flowrate of 30 L/s was able to pass this criterion compared to when the inlets were placed on the same side.



Figure 12. Shows the zone maximum velocity at z = 0.1 m for all cases.

Cases 1, 2, 3, 4, 5, 6, and 8 meet the requirement of not exceeding 0.25 m/s during the cooling season. Particularly, Case 8 is of interest, since by placing the inlets opposite to each other, a case with total flowrate of 50 L/s was able to pass this criterion compared to when the inlets were placed on the same side.

4. Conclusions

This research aimed to investigate the impact of using two inlets instead of one in various configurations for a room equipped with a CIJV. More specifically, the interest was to examine how the flow field in the room is affected by placing the two inlets on the same side of the room vs. placing them on the opposite, and to test these two setups with different flow rates.

In the first part of the results, the velocity profiles show that the highest velocity peak for all cases is obtained at x = 0.5 m and the lowest at x = 3.5 m. The velocity profile development remains similar regardless of the flow rate.

In the zone evaluation, the results show that Cases 1, 2, and 4 meet the requirement of not exceeding 0.15 m/s during the heating season in the occupied zone according the BBR standard. For Case 4, the optimal placement of the inlets is opposite to each other when V = 30 L/s. Cases 1, 2, 3, 4, 5, 6, and 8 all meet the requirement of not exceeding 0.25 m/s during the cooling season. For Case 8, the optimal placement of the inlets is opposite to each other when V = 50 L/s.

Author Contributions: A.A.: Writing—Original Draft, Software, Conceptualization, Methodology, Visualization, Resources; H.Y.: Formal analysis, Review and Editing; T.K.: Supervision, Formal analysis, Review and Editing. All authors have read and agreed to the published version of the manuscript.

Funding: This research received no external funding.

Institutional Review Board Statement: Not applicable.

Informed Consent Statement: Not applicable.

Data Availability Statement: The data presented in this study are shown in the paper.

Acknowledgments: The University of Gävle's laboratory and technical staff for providing assistant with measuring equipment.

Conflicts of Interest: The authors declare no conflict of interest.

Nomenclature

air distribution system		
Boverket's building regulations in Sweden		
computational fluid dynamic		
corner impinging jet ventilation		
displacement ventilation		
heating, ventilation, and air conditioning		
impinging jet ventilation		
mixing ventilation		
Reynolds-averaged Navier-Stokes		
root-mean-square error [%]		
room height [m]		
turbulent kinetic energy [m ² ·s ⁻²]		
pressure $[N \cdot m^{-2}]$		
Reynolds number [-]		
strain rate tensor $[s^{-1}]$		
fluctuating component of velocity $[m \cdot s^{-1}]$		
mean component of velocity $[m \cdot s^{-1}]$		
nominal air velocity of the inlet device $[m \cdot s^{-1}]$		
numerical solution at base grid resolution $[m \cdot s^{-1}]$		
numerical solution at refined grid resolution $[m \cdot s^{-1}]$		
jet velocity normalized by its local maximum velocity [-]		
distance along the diagonal centerline of the room [m]		
cartesian spatial coordinates [m]		
height of the inlet from the floor [m]		
Kronecker delta [-]		
density [kg⋅m ⁻³]		
Reynolds stress tensor $[kg \cdot m^{-1} \cdot s^{-2}]$		
dynamic viscosity $[kg \cdot m^{-1} \cdot s^{-1}]$		
eddy viscosity [kg·m ^{-1} ·s ^{-1}]		

References

- Vimalanathan, K.; Babu, T.R. The Effect of Indoor Office Environment on the Work Performance, Health and Well-Being of Office Workers. J. Environ. Health Sci. Eng. 2014, 12, 113. [CrossRef] [PubMed]
- 2. Kong, X.; Xi, C.; Li, H.; Lin, Z. A Comparative Experimental Study on the Performance of Mixing Ventilation and Stratum Ventilation for Space Heating. *Build. Environ.* **2019**, 157, 34–46. [CrossRef]
- 3. Lee, H.; Awbi, H.B. Effect of Internal Partitioning on Indoor Air Quality of Rooms with Mixing Ventilation—Basic Study. *Build. Environ.* **2004**, *39*, 127–141. [CrossRef]
- 4. Ameen, A.; Cehlin, M.; Larsson, U.; Karimipanah, T. Experimental Investigation of the Ventilation Performance of Different Air Distribution Systems in an Office Environment—Cooling Mode. *Energies* **2019**, *12*, 1354. [CrossRef]
- Ameen, A.; Cehlin, M.; Larsson, U.; Karimipanah, T. Experimental Investigation of Ventilation Performance of Different Air Distribution Systems in an Office Environment—Heating Mode. *Energies* 2019, 12, 1835. [CrossRef]
- Gilani, S.; Montazeri, H.; Blocken, B. CFD Simulation of Stratified Indoor Environment in Displacement Ventilation: Validation and Sensitivity Analysis. *Build. Environ.* 2016, 95, 299–313. [CrossRef]
- 7. Yamasawa, H.; Kobayashi, T.; Yamanaka, T.; Choi, N.; Cehlin, M.; Ameen, A. Applicability of Displacement Ventilation and Impinging Jet Ventilation System to Heating Operation. *Jpn. Arch. Rev.* **2021**, *4*, 403–416. [CrossRef]
- Yamasawa, H.; Kobayashi, T.; Yamanaka, T.; Choi, N.; Cehlin, M.; Ameen, A. Effect of Supply Velocity and Heat Generation Density on Cooling and Ventilation Effectiveness in Room with Impinging Jet Ventilation System. *Build. Environ.* 2021, 205, 108299. [CrossRef]
- Yamasawa, H.; Kobayashi, T.; Yamanaka, T.; Choi, N.; Cehlin, M.; Ameen, A. Prediction of Thermal and Contaminant Environment in a Room with Impinging Jet Ventilation System by Zonal Model. *Build. Environ.* 2022, 221, 109298. [CrossRef]

- 10. Ye, X.; Kang, Y.; Yan, Z.; Chen, B.; Zhong, K. Optimization Study of Return Vent Height for an Impinging Jet Ventilation System with Exhaust/Return-Split Configuration by TOPSIS Method. *Build. Environ.* **2020**, *177*, 106858. [CrossRef]
- Staveckis, A.; Borodinecs, A. Impact of Impinging Jet Ventilation on Thermal Comfort and Indoor Air Quality in Office Buildings. Energy Build. 2021, 235, 110738. [CrossRef]
- Ameen, A.; Cehlin, M.; Larsson, U.; Yamasawa, H.; Kobayashi, T. Numerical Investigation of the Flow Behavior of an Isothermal Corner Impinging Jet for Building Ventilation. *Build. Environ.* 2022, 223, 109486. [CrossRef]
- 13. Hu, J.; Kang, Y.; Lu, Y.; Yu, J.; Zhong, K. Simplified Models for Predicting Thermal Stratification in Impinging Jet Ventilation Rooms Using Multiple Regression Analysis. *Build. Environ.* **2021**, *206*, 108311. [CrossRef]
- 14. Chen, H.J.; Moshfegh, B.; Cehlin, M. Numerical Investigation of the Flow Behavior of an Isothermal Impinging Jet in a Room. *Build. Environ.* **2012**, *49*, 154–166. [CrossRef]
- 15. Wang, P.; Lv, J.; Bai, M.; Wang, Y.; Hu, C. Numerical Investigation of the Flow and Heat Behaviours of an Impinging Jet. *Int. J. Comput. Fluid Dyn.* **2014**, *28*, 301–315. [CrossRef]
- 16. Boverkets Building Regulations—Mandatory Provisions and General Recommendations, BFS 2011:6 with Amendments Up to BFS 2020:4; Boverket: Karlskrona, Sweden, 2011.