American Journal of Engineering Research (AJER)
e-ISSN: 2320-0847 p-ISSN : 2320-0936
Volume-10, Issue-12, pp-34-46
www.ajer.org
Research Paper Open Access

Application of CFD Field to reduce the risk of disease transmission in Small occupancy Ventilated buildings

Jassar M.M. Aljassar[1] and Salem H. Alzaid[2]

[1]*The Public Authority for Applied Education and Training , Vocational Training Institute, Automotive Mechanic Department ,Kuwait.*

[2]*The Public Authority for Applied Education and Training , Vocational Training Institute, Automotive Mechanic Department ,Kuwait.*

[*] *Corresponding Author E-mail: jm.aljassar@paaet.edu.kw*

Abstract *The impinging jet ventilation (IJV) system is an air-distributing strategy to generate a thermal stratification in a room by supplying the air vertically toward the floor, which provides higher ventilation effectiveness than mixing ventilation The impinging jet ventilation (IJV) system is an air-distributing strategy to generate a thermal stratification in a room by supplying the air vertically toward the floor, which provides higher ventilation effectiveness than mixing ventilation The impinging jet ventilation (IJV) system is an air-distributing strategy to generate a thermal stratification in a room by supplying the air vertically toward the floor, which provides higher ventilation effectiveness than mixing ventilation bstract.The impinging jet ventilation (IJV) system is an air-distributing strategy to generate a thermal stratification in a room by supplying the air vertically toward the floor, which provides higher ventilation effectiveness than mixing ventilation. T Abstract.The impinging jet ventilation (IJV) system is an air-distributing strategy to generate a thermal stratification in a room by supplying the air vertically toward the floor, which provides higher ventilation effectiveness than mixing ventilation. To date, however, no simplified prediction model of indoor thermal environment for an impinging jet ventilated room has been established. Therefore, this study aims to propose a simplified calculation model based on "Block Model". This paper first presents a parametric study using Computational Fluid Dynamics (CFD), where total supply airflow rate of a test room is changed as a parameter. Abstract.The impinging jet ventilation (IJV) system is an air-distributing strategy to generate a thermal stratification in a room by supplying the air vertically toward the floor, which provides higher ventilation effectiveness than mixing ventilation. To date, however, no simplified prediction model of indoor thermal environment for an impinging jet ventilated room has been established. Therefore, this study aims to propose a simplified calculation model based on "Block Model". This paper first presents a parametric study using Computational Fluid Dynamics (CFD), where total supply airflow rate of a test room is changed as a parameter. The number of terminalis also changed. Secondly, the paper presents the simplified calculation model of vertical temperature distribution based on block model thatsolves heat flowby advection and diffusion. Here, in this prediction model, the most important parameter is the turbulent thermal diffusivity between two room-space blocks in vertical direction. The room-spaceblocks are classified into two types, i.e., lower and upper part of a room. Based on parametric study using CFD, turbulent thermal diffusivity is arranged forlower and upper block respectively, and it is finally shown that the vertical temperature distribution from block model well agrees with CFD result. The interrupting ventilation (BJV) system is an air-distributing system to generate a current stratification in a room by providing the air vertically in the direction of the floor. A computational revision of air movementfeaturesintimate a ventilated room has been reported in this research. Computation is performed using the commercial code CFD -Task flow, where the air phase is modeled using the 3-D RANS equations with a two equation model for the turbulence closure. The predicted velocities are compared to the available experimental data. For the comparison, computation has also carried out under different operating conditions (air outlet opening location, internal partition, and ceiling obstacle). The results decided that, the wall jet area and recirculation provinces are affected by the location of inlet and outlet openings. Also, the results designate that the model predications are in good agreement with the experiments.*

Keywords*: Numerical model; Air Ventilation; Obstacles; and partition.*

Date of Submission: 02-12-2021 Date of acceptance: 15-12-2021

Nomenclature

A *Supply air area, m^2*
D *pipe diameter, m*
H *Test room height, m*
L *Test room length, m*
M *Mass flow rate, kg/s*
Re *Reynolds number*
R *Inlet and outlet air opening radius, m*
T *Air temperature, ^oC*
U *Total velocity in ventilated section m/s*
W *Test room width, m*
X, Y, Z Cartesian coordinates.

I. INTRODUCTION

An extraordinary viral disease has brought our globe to a halt, impacting most of mankind's activities. Now, nearly more than 620,000 people are dead all-inclusive, the universalreduced is on the verge of an unparalleledsadness, with the COVID-19 sicknessquietrampant and a second wave expected as unavoidable. COVID-19 goes to the collection of coronavirus, also identified as Spartan Acute Breathing Syndrome Corona Virus 2 (SARS-CoV-19) **[1]**.**T. Lipinski et al. [1]** decided that many of the current ventilation strategies that rely oncentralized air distribution and ceiling level supplyor recirculation can provide the optimum conditions for rapid diseasespread in high occupancy buildings. On the additionalinfluence, transpositionaerationplans, such as the normalairing or obviouslysupportedfresheningdiscovered above, can afford an actualfirstidea for recovering our constructions for harmless use. If the outdoor conditions are favorable (e.g., temperature differences and wind patterns), naturally ventilated buildings have higher ACH than mechanically ventilated buildings.

Air ventilation becomes more important in our daily life because of increasing hazard of pollutants. The most important requirements are removing heat generated by different sources such as, sun, lighting, and other electrical devices. The air supplyorganization must also supply fresh air to the room. Also, air must be distributed with regard to the heat sources in order to obtain a uniform thermal condition in the occupied zone. Ventilation is necessary in order to prevent unduly high concentrations of carbon dioxide, moisture and to remove bacteria, odor, products of combustion, etc. More recent studies showed that supplying fresh air in the operating room is an adequate way to remove bacteria generated. Today air ventilation becomes very important because people spent most of their interval in interior environment. In modern design, an open space is divided into offices using internal partitions. **Wu, and Chen. [2]** decided that the interior air eminence may influenced by inlet/outlet relative locations larger than increasing the fresh air supply rate. The high efficiency ventilation systems are needed for most health care facilities, especially in high clean standard required areas, such as operation rooms. However, the great air adjustmentamount does not guarantee the high ventilation efficiency of a ventilated space. They also reported that basically, the airflow pattern will also play an significantpart in obtaining high ventilation efficiency in a ventilated space. One general method to improve indoor air quality is to simply increase the ventilation rate.

K. S. Mushatet[3] showed that the barrier jet section and recirculation areas are precious by both the place of inlet and outlet openings. Similarly the recirculation regions are increased with increasing Reynolds number and decrease with decreasing aspect ratio. **J.D. Posneret al.[4]**indicated that the mathematicalimitations and velocimetry extents show how impediments can significantlyimpact the air flow and impuritytransference in a room. It is significant, that barricades be considered in freshening design. Simulations expect the measured tendencies in a typicalchamber very well, with absolute errors not greatlyhigher than 20%. In this study, the RNG model most exactlyexpects the flow in a subdivided room, taking the gross effects of a large flow impediment.**G. Zhang, et al.[5]**established that a solid impedimentsituated upstream of the exposed sidewall of the scale model exaggerated the velocity and chemicalabsorption field in and round the building model. Also observed that the **o**bstacle located three intervals the edge height (3H) upstream from the exposed sidewall, the velocity field improvedgreatly compared with the case with no obstacle. Exchanging airflow arrangementsdirected to amplified CO_2 concentration among the obstacle and the windward sidewall and lower CO_2 concentration downstream of the building. Numerical imitations were achieved with a full-scale construction. The simulation consequencesshowed the same trends for both the velocity field and CO_2spreading. The simulated CO_2 concentrations downstream of the building were in fair contract with the measured results at heights below 1H.

K.c. Chung [6]suggested that theexperimentation is approvedthat in a full-scale conservational control space. The arrangement between the calculated and the measured effects of temperature and velocity fields are rather well. He also showed that expectation results of pollutant particle's course will give valuable evidence to be used for assessing the indoor air excellence in the design technique.

B. k. Morsy et al. [7] decided that in the present study comparison of relatively conventional three dimensional computational liquid dynamics (CLD) results with measurements of air movement in an isothermal scale model.

J. Posneret al [8]reported that in pretending the flow in the model chamber, and outcomes of the numerical replications and velocimetry extents show how obstacles can significantlyaffect the air flow and impuritycarrying in a room. It is important, hence, that impediments be considered in ventilation design. Imitations predict the measured trends in a typicalapartment very well, with relative errors not much higher than 20%. In this study, the RNG model most exactlycalculates the flow in a separated room, taking the gross properties of a large flow obstacle.Numerical additions were carried out by using CFD-TASC flow in**[9]** code. This code solves the Reynolds averaged Navier–Stokes equations in originaladaptable form. The standardk–εinstabilityideal used to study the effect of turbulence of fluid flow. Also, in demand to revision the streamclose wall; wall purpose is recycled to decide the wall streams. This program uses a determinatecomponent based finite volume method, to realizea convergent result. An expression to evaluate the ventilation of the diffusion system as a temperature ratio form is presented in **Ramadan B.M [10].**

W. A.Aissa et al. [**11**] reported that mathematicaldesigns are achieved by Lagrangian-particle tracking model for expectingelementflights of detachedsection, and typical k-ε model for predicting the windy gas-solid movements in corners. Comparisons made between the imaginaryoutcomes and investigationaldocumentsaimed at the rapiditypaths and elementflights show good agreement.

Ventilation as the means of slowing down infection rate

ASHRAE standard [12]explainsthat suitable Indoor AirQuality (IAQ), as "thereare no recognizedpollutantsin the air at destructiveapplications, and by which anextensivepopular (80% or more) ofthe personsshowing do not precisedispleasure.

II. NUMERICAL TECHNIQUE

S. J. Rees, and P. Haves[13]investigated that. Imaginingexperimentations, velocity extents and correlatedmathematical studies specified that with more heat gains the columns have enoughmotionto drive flow across the ceiling surface and down the walls. The consequence of required, as opposite to normal convection, is also recommended by rather low Richardson Number (Ri) values found near the upper limit. Additionally, in cases by means oftemperately high interioradditions, assessment of the hotnessgradesshowed that the influence of ceiling surface high temperature on the gradation of mixing and the degree of the heatgrade were of lessersignificance. **Chen et al. [14]** reported that the air feature and currentluxury of the kitchen environment by employing a finite volume CFD code. They studiedthe properties of depletecapacity on the particulate matter movement, age of the air, and comfort condition indices.

L. Tian et al.[15]demonstrated that the concentration is at the same level when the contaminant source locates up-steam to the occupant. The concentration in the workingarea (<1.9 m from the floor) is also lower when the contaminant source detects on the bottom. At stream air hotnessenhanced for transposition ventilation, the toluene concentration in the breathing region for layeraeration is higher than that for transposition ventilation when the area source locates on the four surrounding walls of the room.

H. Qian and X. Zheng [16]decided that aggregateaeration rate is alleged to decrease thecross infection of airborne transmitted diseases by eliminating or reducing pathogen-ladenflying droplet focuses. A greateraerationlevel can insipid the polluted air inside the space more fast and decline the hazard of cross impurity. They initiate that the tuberculin editionamongst HCWs was intenselyrelated with insufficientaeration in general long-suffering rooms and period of work.

The purpose of the present work is to study the influences of some main parameters (air outlet opening location, ceiling obstacle and internal partition) on the air movement in a ventilated room in additional to evaluate the effectiveness with which CFD can capture flow details near substantial flow obstructions, since this capability helps determine the usefulness of as a ventilation design tool and to predict flow details.

III. STUDY CASES

A model of air ventilated room of 0.92 m long and 0.62 ×0.62 m cross section was used for computational model. The test room is provided by only one inlet air opening, and four preparedness to fight airborne disease is essential and SARS-CoV-2 offers the chance to research and apply mitigating ventilation solutions which could prove life-saving now, and in the future when another virulent and deadly pandemic arises [13]air outlet opening. Three openings are at the rear wall and the fourth one at the front wall below the inlet opening. A ceiling obstacle was mounted tightly to the ceiling at a distance of 0.5 m from the air inlet opening. Also an internal partition (0.32 × 0.42 × 0.07 m) was fixed well at a distance of 0.46 m from inlet as in **Figure 1**. The inlet and outlet opening air locations are shown in **table 1**

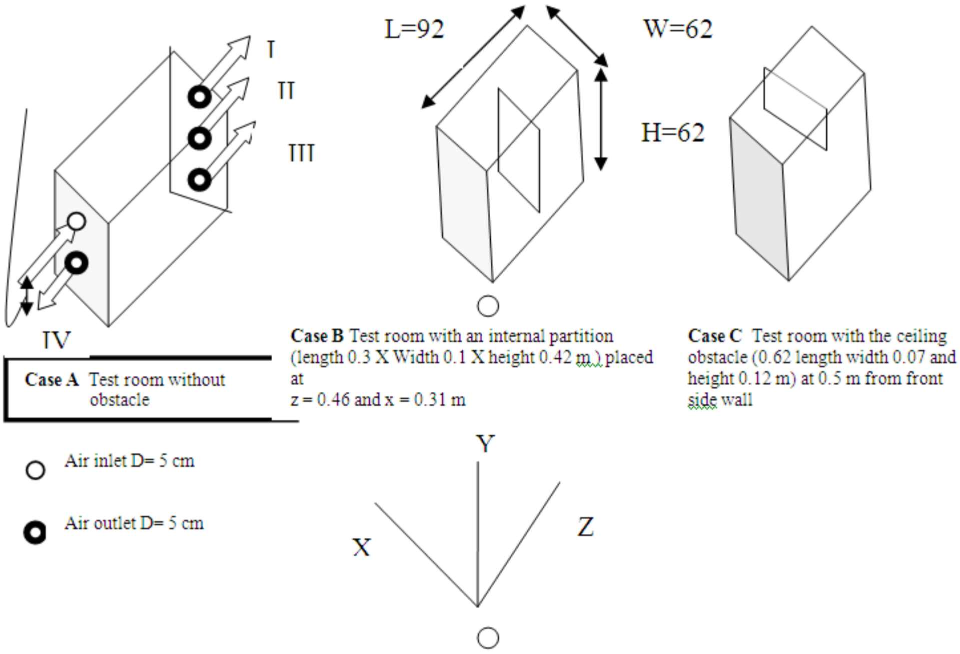


Fig. 1 Schematic diagram of Physical model

Fig. 2 Computational model

Open	Y
Inlet	50.84
IV	10.84
III	7.254
II	31
I	50.84

Table 1 Opening Air Location

Six runs were carried out using a commercial code CFX-TASC flow as in table 2 to study effect of air outlet location, internal partition, and ceiling obstacle on space flow characteristics inside a ventilated room.

Run No	Outlet opening	Inlet air velocity m/s	Inlet air temperature °C	Internal partition	Ceiling obstacle
1	I	36.36	38.5	without	without
2	II	36.36	39.5	without	without
3	III	36.36	38	without	without
4	IV	36.36	35.5	without	without
5	IV	36.36	39	with	without
6	IV	36.36	39.5	without	with

Table 2 Cases study

IV. RESULTS AND DISCUSSIONS

The analysis and discussion include the influences of location of outlet opening, an internal partition and a ceiling obstacle on the velocity pattern, raging kinetic energy and also on the temperature contour. Finally a comparison between the predicted velocity and previously measured was done.

4-1 Effect of Outlet Position on Velocity Vector a cross the Ventilated Room.

Figure 3 (a,b,c, and d)reported the effect of outlet position (outlet **I,II**, **III**, and **IV**) on the velocity vectors across a ventilated room at an average Reynolds number of 67000. **Fig.3** indicated that changing the outlet position has remarkable influence on the jet behavior inside the room.Also, it concluded that, as the outlet position far from inlet axis the jet spread angle increase this mean that outlet **III** has greater spread angle than outlet **II** and **I.** It is evident that the velocity is high as close to the ceiling from inlet and will be reduced as closed to the floor**[2, 3].** It could be note that a re-circulating flow is predicted near the outlet and corners. It could be reported that the circulation zone will be greater as the outlet is closes to the inlet axis, which mean that outlet **I** has greater circulation zone than outlet **II** and **III** [**16**]. In case **I**,**Figure 3a**, when the air be far from the inlet the inertia force is decreased and the flow begins to form a semi stagnant core region. However the stagnant core region is not uniform. In the case **IIIFigure 3 c** a noticeable decrease in the axial velocity occurs in the reattachment zone followed by a significant increase in the wall jet area[**16**]. For the case (outlet **IV**) **Figure 3.d, a** massive recirculation zone is predicted near the rear wall. The flow from opening supply may be divided into three regions; the circulating flow in the corners, the central core region and the wall jet region.

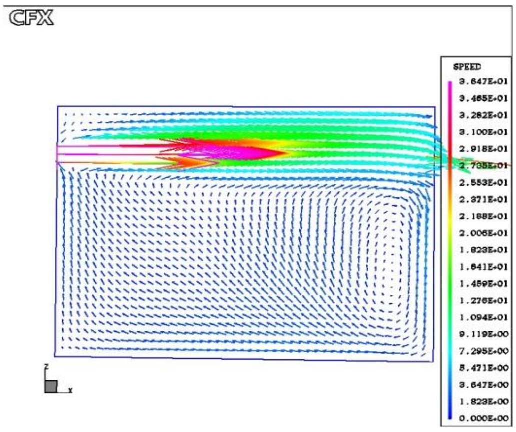


Fig. 3a

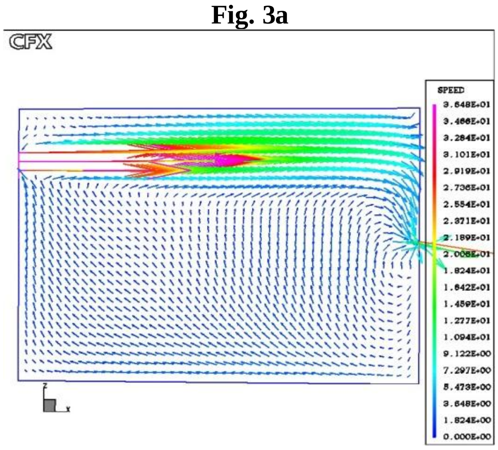


Fig. 3b

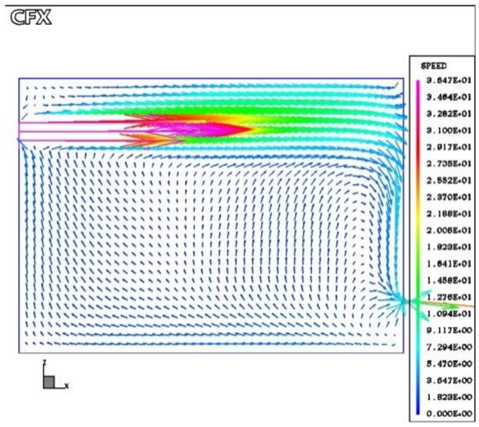


Fig. 3c

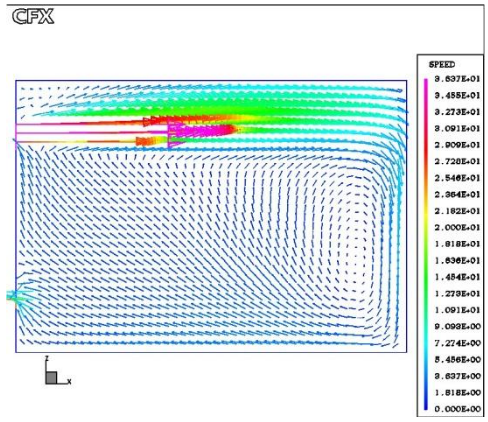


Fig. 3d

Fig. 3 Velocity vectors for different inlet and outlet apertures, (a) outlet I, (b) outlet II, (c) outlet III, and (d) outlet IV

4-2 Effect of Outlet position on a turbulent kinetic energy cross the ventilated room.

Figure 4 shows the spreading of windykinematic energy for thestudied cases. It can be proved that location of apertures affects the distribution and principles of instability kinetic energy. The relatively high values of stormykinematic energy are found in case **(c).**

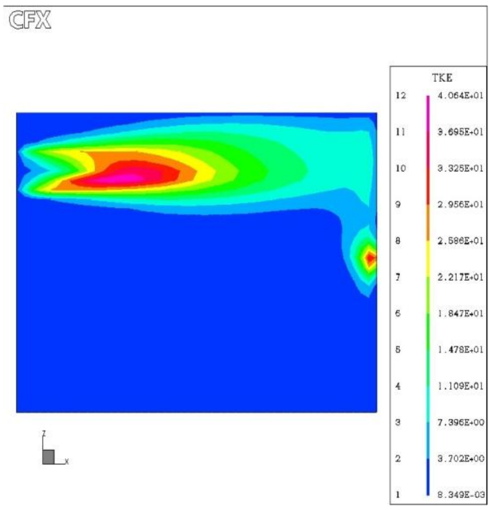


Fig. 4a

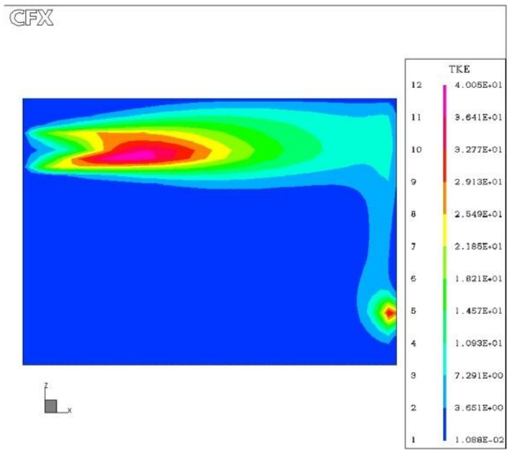


Fig. 4b

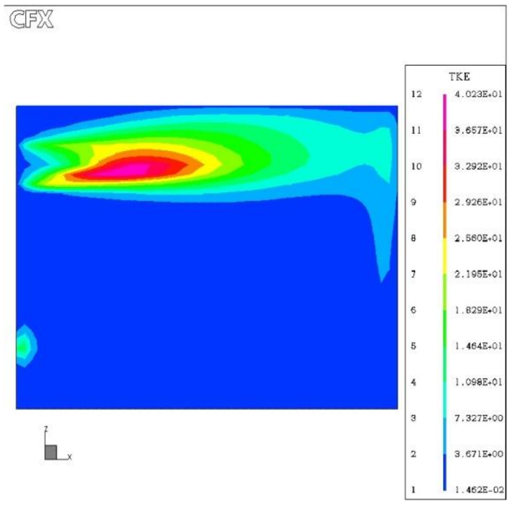


Fig. 4c

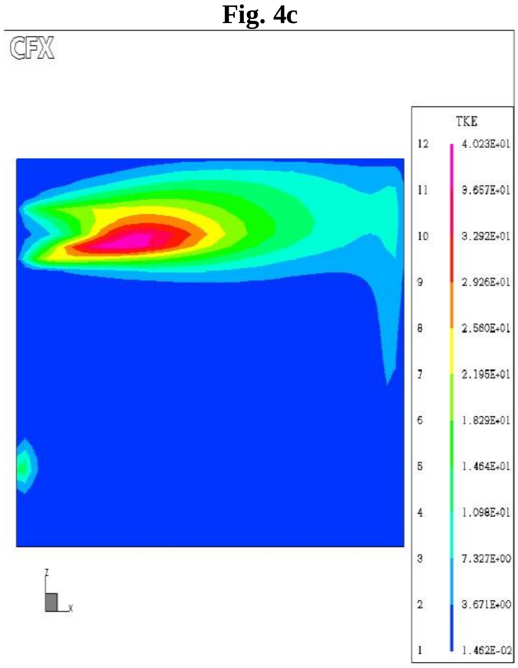


Fig. 4d

Fig. 4 Contours of turbulence kinetic energy, (a) Outlet I, (b) Outlet II, (c) Outlet III, and (d) Outlet IV

4-3 Effect of Outlet Position on Temperature Contour A cross the Ventilated Room.

Figure 5 showsthe effect of outlet position (outlet **I,II, III**, and **IV**) on temperature field a cross a ventilated room at Reynolds number of 67000. It is noticed from the Figure that changing the outlet position will have slightly remarkable influenced on the temperature contour inside the room because the isothermal state is considered, also the figure shows that the temperature is higher through the jet axis and decrease toward ceiling and floor. Also it could be note that there is a zone in the occupied space could be lower in temperature as it will be far from recirculation closed to corner [**14**].

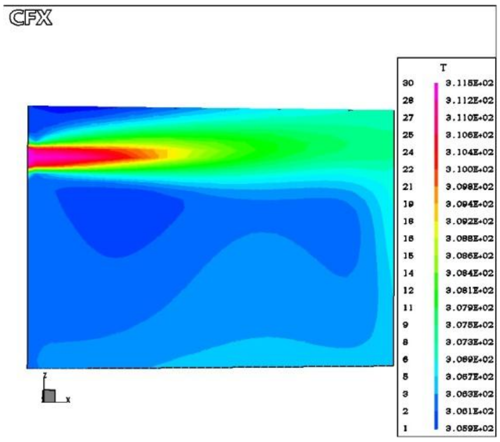


Fig. 5a

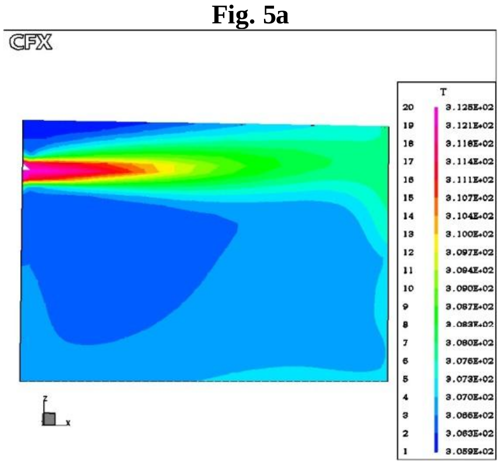


Fig. 5b

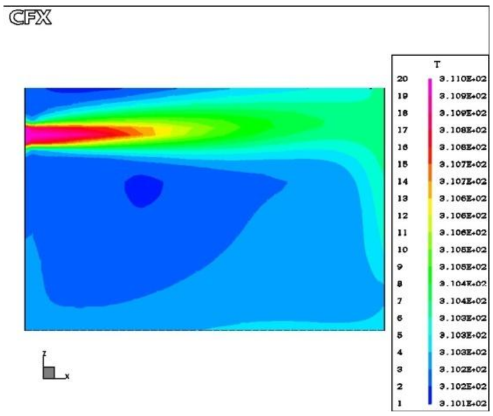


Fig. 5c

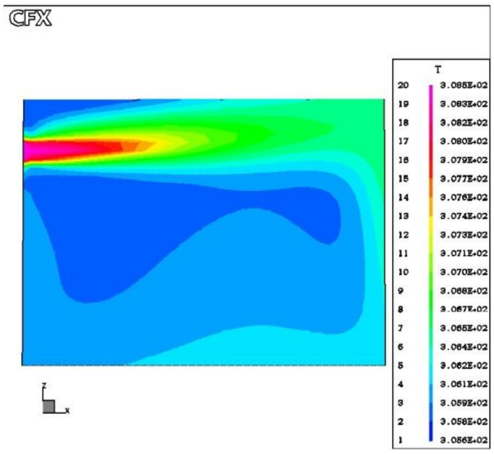


Fig. 5d

Fig. 5 Temperature field with different inlet and outlet positions, (a) outlet I, (b) outlet II, (c) outlet III, and (d) outlet IV

4-4 Effect of floor partition on velocity vector and temperature contour.

Figure 6illustrates the effect of floor partition on the velocity vectors. It could be seen that the flow configuration is affected by the floor partition inside the room. Secondly the velocity will be slightly higher above the floor partition, and this could be explain that the partition reduces the flow area, and as flow rate is constant so the velocity will be higher as shown in **Fig. 6 a.** From the **Fig. 6 a** the floor partition divided the room into two zones one between partition and rear wall, which has more ventilated (as the velocity is higher due to blockage in cross sectional area of room) than the other zone between partition and front wall (as partition make resistance for air). The **Fig. 6a**show also that the generation of vertical structures as the flow crosses over the partition. Besides producing rotational flow patterns, the partition also creates a stagnant flow region on the lower half of the second zone. Finally it could be evident from **Fig. 6** that there are low velocities in the workingregion in **Fig. 6 a** as compared with **Fig. 6 b (**without partition) and this could be explained because the internal partition resists the air circulating jet inside the room**[8].**

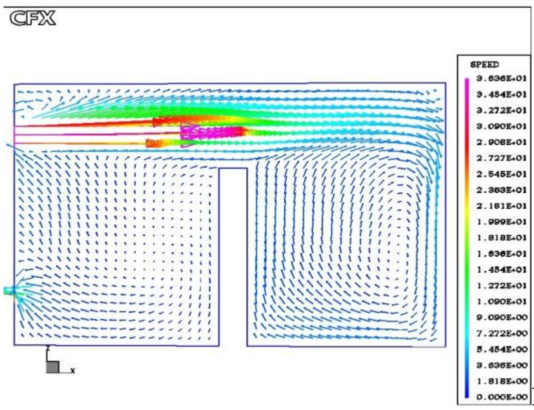


Fig. 6a

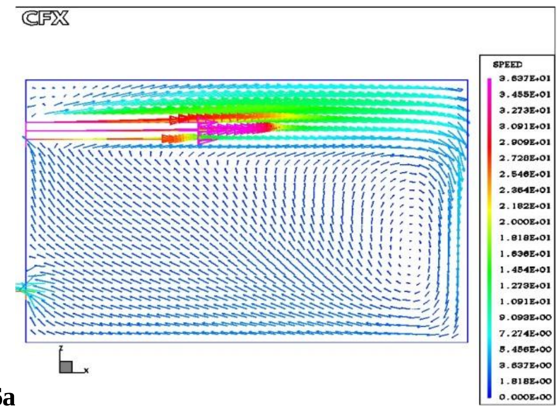


Fig. 6b

Fig. 6 Indoor air flow pattern with (velocity vectors), for Outlet No **IV** with and without floor partition (a) with floor partition (b) without floor partition

Figure 7 shows the effect of floor partition on temperature field. It could be seen that the temperature configuration is affected by the floor partition inside the room**[8].** Secondly the jet regions and zone between partition and rear wall has a slightly higher temperature above the regions between partition and front wall and this could be explained as jet region and region between partition has a higher velocity which mean higher kinetic energy which dissipation into room and converted to thermal energy in form of temperature more over the region between partition and front wall is similar to case without partition as in **Fig. 7 b**

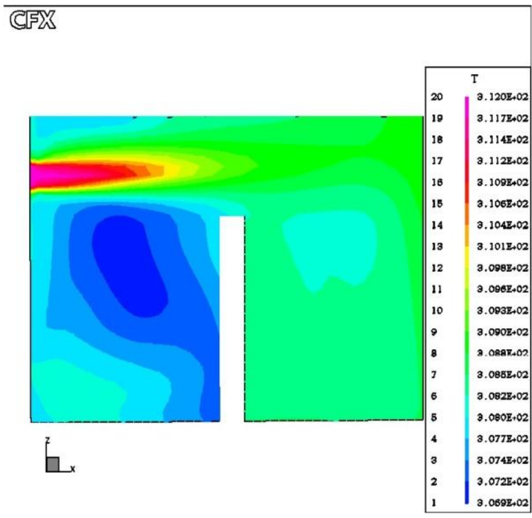


Fig. 7a

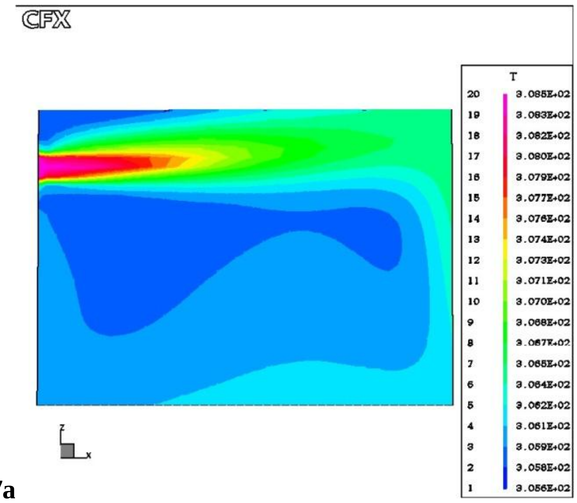


Fig. 7b

Fig. 7 temperature field, for outlet No IV with and without floor partition (a) with floor partition (b) without floor partition

4-5 Effect of Ceiling Obstacle on Velocity Vector and TemperatureContour.

Figures 8 and 9 showsthe effect of aupper limit obstacle on velocity vector and temperature field. It could be seen that the flow velocity and temperature with ceiling obstacle is approximately look like flow velocity and temperature without ceiling obstacle except that the ceiling obstacle deflect the jet. More over the region between ceiling obstacle and rear wall has low velocity and low temperature and this could be explain due to resistance of ceiling obstacle to flow [**16**]..

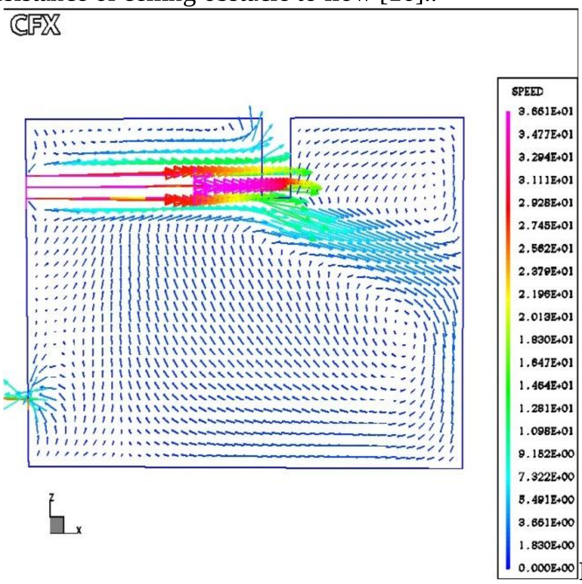


Fig. 8a

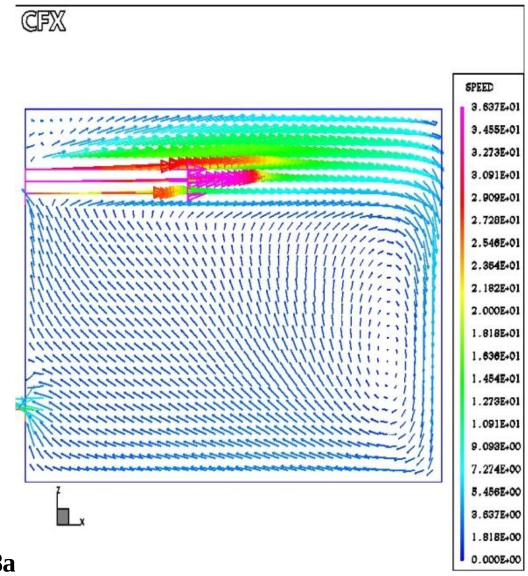
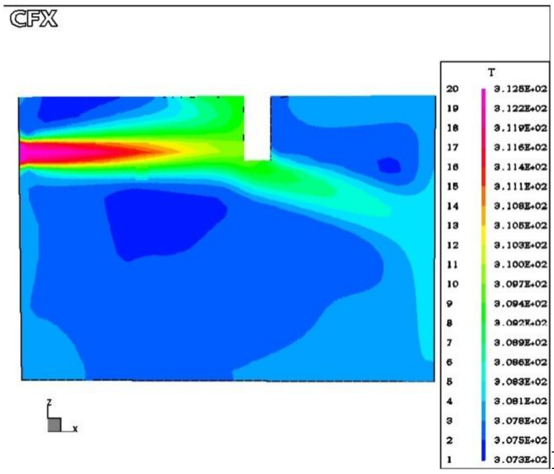


Fig. 8b

Fig. 8 velocity vector, for outlet No **IV** with and without ceiling obstacle(a) with ceiling obstacle, (b) without ceiling obstacle.



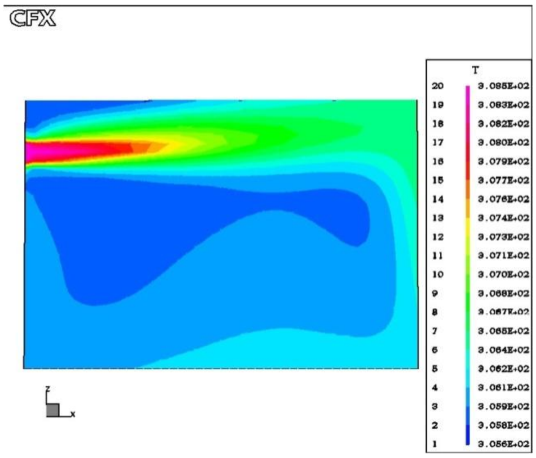


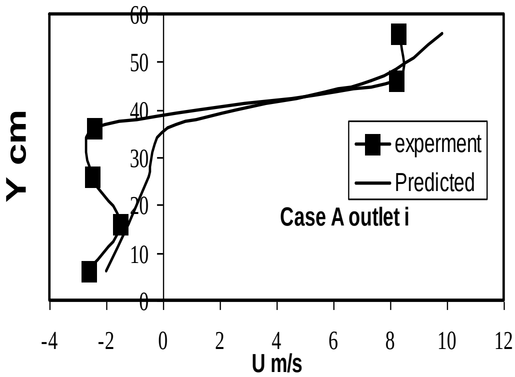
Fig. 9a

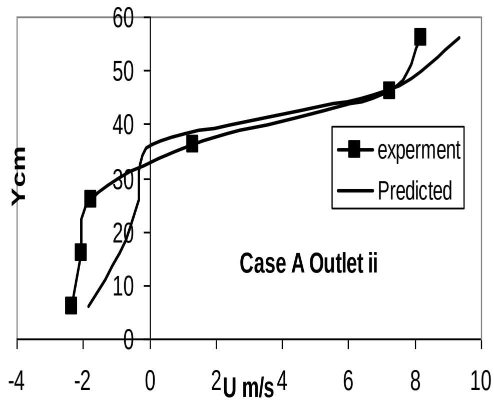
Fig. 9b

Fig. 9 Temperature field, for outlet No **IV** with and without ceiling obstacle (a) with ceiling obstacle, (b) without ceiling obstacle.

4-6 Comparison between predicted and measured velocity

Figure 10 shows Comparison between experimental and predicted local velocity across vertical distance for X=65.72cm, Z = 31cm, and Re =67000 at different conditions (outlet position, floor partition, and ceiling obstacle). It could be seen that the predicted value agrees with the measured value**[4, 6, 7].** And most of deviations are coincident at reverse flow regions and for floor partition. While there are some key regions of disagreement between the numerical simulation and the measurements, there are also aboard regions of agreement**[4, 6, 7].**





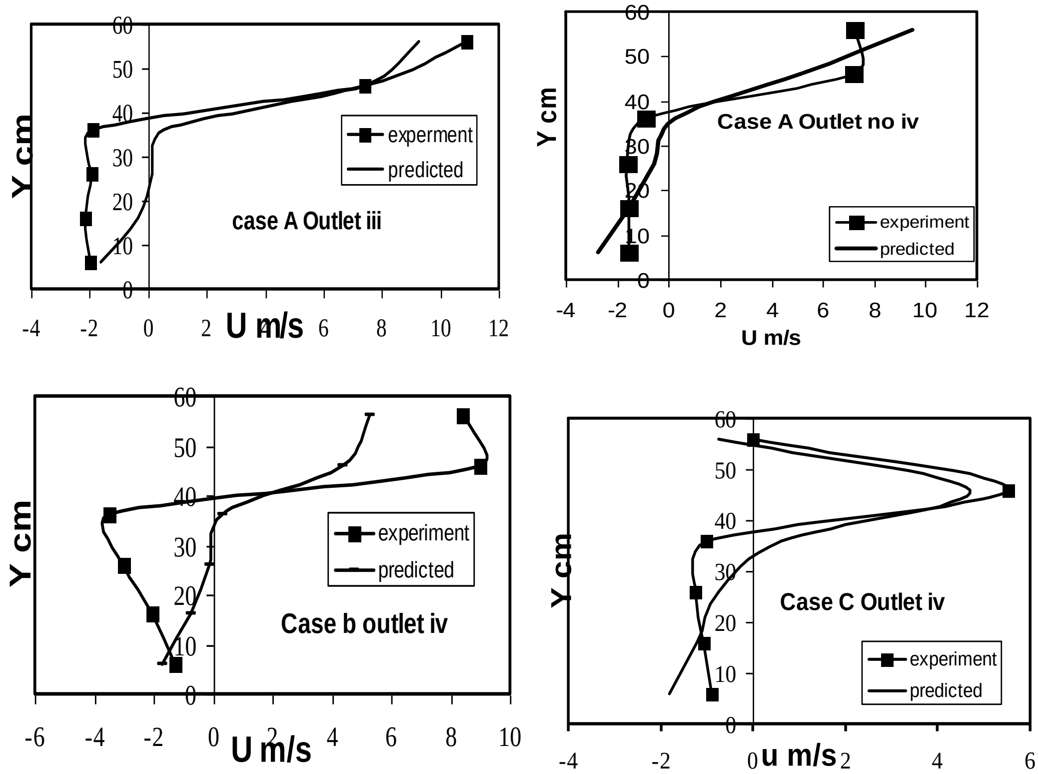


Fig. 10 Comparison of the results predicted by the present studywith published experimental results.(ref.7) local velocity across vertical distance for z=65.72 cm and x = 31 cm, and Re =67000

V. CONCLUSION

Building Services of high occupancy buildings must be better adapted as a substance of determination to facilitate the reduction of disease transmission resulting from inappropriate or inadequate ventilation. The COVID-19 pandemic has exposed areas requiring urgent development to protect both our health, wellbeing and the economy by providing safe indoor environments for employees or students. The numerical results showed decentcontractto previous experimental ones with some deviations in some regions**[1].** The following main points could be drawn from the present numerical study on the influenced of air outlet position, partition floor, and ceiling obstacle on room air distribution.

1- Velocity pattern and temperature field inside the ventilated room is influenced by an air outlet location**[14].**

2- Air velocity inside the engagedarea is high inside the room without obstacles, however the velocities of the reverse flow are reduced inside the room with internal partition**[8, 15]**.

3-The reverse flow region depends strongly on air outlet location, however as air outlet opening is moved far from the ceiling, the reverse flowzone decreases except in case of outlet opening No IV**[2, 3].**

4- Ceiling mounted obstacle sometimes deflects air jet into the occupied zone, so the air haste in the busyregion increases, but the air jet velocity values slightly decrease**[8]**. This is suitable foradapted as a trouble of insistence to facilitate the reduction of disease transmission [**13,17**] **.**

5-Ceiling obstacle leads to a circulating flow zone close to the ceiling**[8].**

REFERENCES

[1]. T. Lipinski, D. Ahmad, N. Serey, and H. Jouhara "Review in ventilation plans to decrease the hazard of illness spread in great residence structures", International Journal of Thermo fluids, Vol. 7–8, Nov. 2020, 100045

[2]. Wu, Y.L. and C.W. Chen,. "The influence of indoor obstacles on general ventilation designs", J. Applied Sci., 2010 10: 2855-2861.

[3]. Khudheyer S. Mushatet "Numerical Simulation of Turbulent Isothermal Flow in Mechanically Ventilated Room", JKAU: Eng. Sci., Vol 17 No. 2, pp: 103 - 117 (2006 A.D. /1427 A.H.).

[4]. J.D. Posner, C.R. Buchanan,and D. D. Rankin, "Quantity and calculation of interior air movement in a typical apartment", Energy and buildings 35 (2003), 515-526.

[5]. G. Zhang, A. Ikeguchi, J. S Strøm, S. Morsing, H. Takai, P. Ravn, and L. Okushima, "Impediment Special effects on Airflow and Pollutant Spreading about a Obviously Examined Cattle", Building Agricultural Engineering International: Vol.V (2003).

[6]. K.c. Chung " Three dimensional analysis of air flow and contaminant particle transport in a partitioned enclosure" Building and Environment 34 1999 7-17
[7]. Abel Tablada, Bert Blocken, Jan Carmeliet, Frank De Troyer,Han Verschure "The influence of courtyard geometry on air flow and thermal comfort: CFD and thermal comfort Simulations.
[8]. B.K.Morsy, M. R. Diab and S.A. Ahmed "Effects of air outlet position, floor partition and celling obstacle on air ventilation inside a room" Journal of engineering and applied science Bulletin of the Faculty of Eng. – Cairo University, 2001, Vol. 48, No1,feb 2001, pp 155-174.
[9]. J. Posner, C.R. Buchanan, and D. Dunn-Rankin, "Extent and calculation of interior air current in a typical chamber", Energy and Buildings 35(5):515-526, DOI:10.1016/S0378-7788(02)00163-9,
[10]. ASC. CFX-TASC flow documentation version 2.9.0, 1999 (Advanced Scientific Computing, Ltd,Waterloo, Ontario,Canada).
[11]. Ramadan B.M. "Air flow characteristics of a psychometric test chamber with variable ventilation openings" M.Sc. Thesis, Minia University, 1993.
[12]. W. A. Aissa, T. A. Mekhail, S. A. Hassanein, and O. Hamdy," Experimental and Numerical Study of Dilute Gas-Solid Flow inside a 90° Horizontal Square Pipe Bend", CFXTASC Flow Version 2.9 Theory Documentation," AEA Technology Engineering Software Limited, Waterloo, N2L 5Z4, 1999.
[13]. ASHRAE Standing Standard Projecrt Committee 62.1, ANSI/ASHRAE Standard 62.1-2019.
[14]. S. J. Rees, and P. Haves, "An experimental study of air flow and temperature distribution in a room with displacement ventilation and a chilled ceiling", January 2013, Building and Environment 59:358–368
[15]. Z. Chen, J. Xin, P. Liu,"Air quality and thermal comfort analysis of kitchen environment with CFD simulation and experimental calibration", Build.Environ., 172 (2020), Article 106691
[16]. L.Tian, Z. Lin, and Q. Wang, "Assessment of smoky impurity dispersion below layer aeration and underneath translation aeration", Building and Environment, Vol. 45, Issue 9, Sep. 2010, Pp 2035-2046
[17]. H. Qian, X. Zheng,"Ventilation control for airborne transmission of human exhaled bio-aerosols in buildings", Vol. 10, July 2018: Journal of Thoracic Disease.

Jassar M.M. Aljassar, et. al. "Application of CFD Field to Reduce the Risk of Disease Transmission in Small Occupancy Ventilated Buildings."*American Journal of Engineering Research (AJER)*, vol. 10(12), 2021, pp. 34-46.