Journal of Computational Applied Mechanics 2021, 52(3): 423-432
DOI: 10.22059/jcamech.2020.292518.455

RESEARCH PAPER

Effect of wind velocity and flare height parameters on pollution dispersion from one flare with zonal method

Alireza Sayar, Ali Falavand Jozaei*

Department of Mechanical Engineering, Ahvaz Branch, Islamic Azad University, Ahvaz, Iran

Abstract

With increasing growth of industrial units in developing countries and pollutants produced by these units, nowadays distribution and dispersion modelling of atmosphere pollutants especially in urban areas is an inevitable importance. Dispersion modelling of atmosphere pollutants is a methodology to estimate focus and concentration values of pollutants related to emission source in different seasons. In current study, using numerical analysis, a thermal diffusion of flare and focus values of industrial pollutants simulation using air zonal methodology has been presented. After studying pollutants emission in open area, validation has been done using laboratory data results and computational fluid dynamics method. The results of this study indicate the ability of presented air zonal methodology to predict thermal diffusion of flare and distribution concentration of pollutant source and information gained from this analysis. Then an exploration of effective parameters in pollution emission such as wind velocity, flare height, and pollution emission rate in downstream has been done. As the results show, when wind velocity rises by 130%, pollution will reach far away from production source and with increase in flare height by 25%, the pollution concentration values on the ground has been reduced by 44%. Also with addition of barrier in pollution dispersion path, pollution level will increase by 60%.

Keywords: Flare, Zonal methodology, Pollution diffusion, Special cells

Introduction

Air zonal method is a model between computational fluid mechanics and one node model. This method is capable of simulating different phenomenon that can't be reviewed with one node models; such as pollution distribution, heat saving in wall, Asymmetric thermal radiation and heating and cooling in total. One node and multiple zone methods cannot study many issues since they consider thermal and pollution distribution to be steady in each zone. Air zonal method is based on classifying space into multiple subdomains (cell) which assumes that thermal, humidity and pollution concentration in each cell is homogenous. Air zonal method can be used to model inner and outer media. Recently, this method is used to study mechanical and natural ventilation system performance, air parameters distribution prediction and pollution distribution [1].

Air zonal method allows us to accurately calculate physical parameters in each domain to achieving details including air quality and energy analysis. Air zonal method improves the full mixing assumption which is used in one node and multiple zones methods. Because of its low calculation time, this method is a suitable method for predicting air flow in zone to calculate desired parameters.

* Corresponding Author. *Email Address*: falavand@iauahvaz.ac.ir

In this study, emitted pollution in the air caused by a flare in sample space with zonal method is modeled and simulated, then obtained results is validated with the results of computational fluid dynamic method.

Theory and historical background

Momentum zonal model assumes that velocity in standard cells is low enough that ignores momentum. In momentum zonal model equations it's assumed that velocity and viscosity is low enough that flow is considered inviscid flow and an investigation on momentum conservation has been performed. In this model large meshes with finite volume numerical technic is used to solve Euler equation.

Equations of this model is between Bernoulli equations, pressure zonal model and computational fluid dynamics models. In modeling there's no need to solve flow field completely and solving until reaching energy balance equation is enough. In room temperature usually turbulent flow exists too, and this model and models that are based on air being inviscid don't consider viscosity and therefore turbulence flow; so in issues that turbulence flow is important, this model can't be used. Momentum zonal model is used to compare results of zonal models mostly.

Voeltzel [2] used an air zonal method to predict temperature field and air flow in a building. He also considered thermal radiation exchange into building. To collect experimental results, he used a building with 5.1 meter height and measured the temperature on a line in the middle of building for 56 hours in 4 different points with different heights. Numerical solution results had acceptable conformity with the measurements.

Inard and Buty [3] validated Lebrun's model with multiple experiments and measuring temperature distribution in a controlled environment.

Wurtz [4] and Bouia [5] presented a type of air zonal method in which with solving pressure field, temperature distribution and air flow is calculated for inner spaces. In Wurtz's model, energy and mass balance in each zone will be written and mass flow rate is calculated for common surface in each two zones. He validated his model by comparing different experiments and other numerical solutions. Simulation results of Thermal source effect had good conformity with experimental results.

To calculate thermal load and energy consumption in building, based on multiple zones models which assume full mixing of air in room, non-uniform distribution effects of room air temperature are not considered. To solve this problem, Chen et al (1998) [6] presented a new zonal method to calculate load and energy consumption of building by predicting air flow and temperature distribution in inner environment. This model was validated with experimental results and numerical solutions with computational fluid dynamic.

Air zonal method, have been developed to compare thermal system performances for many years. Studies in this context, includes water radiator [7-9], electrical heaters [10] and radiation panels performance review [10], ceiling heating [10], floor heating [11] and thermal pumps [12].

Huang and Haghighat (2005) presented a 3 dimensional zonal method with air jet and source and endpoint pollution, to predict air flow distribution and pollution concentration in room. To validate this model, air flow distribution in a room with mechanical ventilation was compared with results of computational fluid dynamic method. Temperature distribution with experimental results and pollution concentration distribution with computational fluid dynamic method was compared. Results indicated that this model with large meshes can present enough data for air, temperature and pollution distribution in room [13].

Materials and methods

In environments that air flow is created only by free displacement and it has low air velocity, it is possible to simplify momentum equation in a way that no significant error will be made. For control volume shown in figure 1, momentum conservation Discretization equations in x, y and z axis (gravity in –z) are as below [14]:

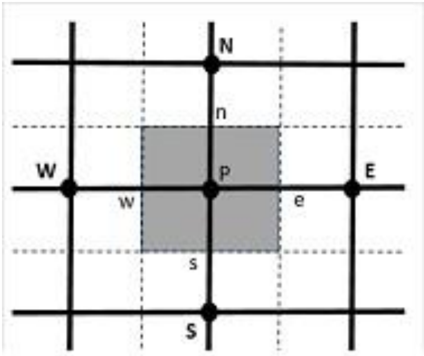

Figure 1. Sample control volume [13].

$$\left(\frac{1}{2}\rho_e u_e\right)u_e = \varepsilon_u C_{d,u}^2 (P_P - P_E)$$
$$\left(\frac{1}{2}\rho_n v_n\right)v_n = \varepsilon_n C_{d,v}^2 (P_P - P_N) \quad (1)$$
$$\left(\frac{1}{2}\rho_t w_t\right)w_t = \varepsilon_w C_{d,w}^2 \left((P_P - P_T) - (\rho g h)_{w,t}\right)$$

In which $C_{d,w}$, $C_{d,v}$ and $C_{d,u}$ are as below:

$$C_i = \sqrt{\frac{\left|\frac{1}{2}\rho_i u_i A_i\right|}{\left|a_i - \sum_{nb} a_j \frac{u_j}{u_i}\right|}} \quad (2)$$

Therefor mass flow rate $\dot{m}_{i,j}$ crossing from common surface of i and j cells is dependent on average pressure difference, common cross section between them, air density and an experimental constant (C_d). So mass flow rate crossing from j to i zone with a common vertical boarder can be defined as below:

$$\dot{m}_{i,j} = \varepsilon_{i,j}\sqrt{2\rho_i} \times C_d \times A_{i,j}\left|p_i - p_j\right|^{0.5} \quad (3)$$
$$\varepsilon_{i,j} = sign\left(p_i - p_j\right)$$

To take account effects of height on cell head pressure, hydrostatic pressure difference for horizontal cells with vertical flow must be inserted in pressure terms, so mass flux term for cells with common horizontal boarders is calculated from equation 4 [15]:

$$\dot{m}_{i,j} = \varepsilon_{i,j}\sqrt{2\rho_i} \times C_d \times A_{i,j} \times \left| p_i - p_j - \frac{1}{2}(\rho_i g h_i + \rho_j g h_j) \right|^{0.5}$$
$$\varepsilon_{i,j} = sign\left(p_i - p_j - \frac{1}{2}(\rho_i g h_i + \rho_j g h_j) \right) \quad (4)$$

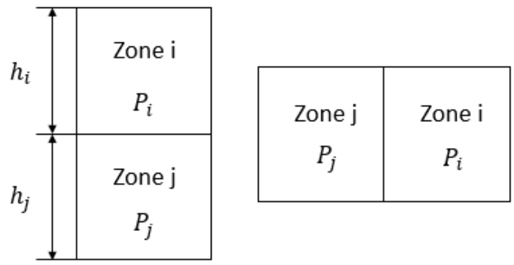

Figure 2. Flow between two zones with common vertical and horizontal surfaces [14].

So with writing mass conservation equation for desired control volume, we can calculate pollution concentration.

$$\sum_{j=1}^{n} \dot{m}_{i,j} + \dot{m}_{source} = 0 \quad (5)$$

It can be seen that to solve unknown pressures, mass conservation equations will turn into system of non-linear equations. To solve this system, system of non-linear equations solution methods like newton method (which is practically a type of Linearization) can be used. In this study we used newton method.

Mass transfer equation governing gas pollution dispersion phenomena can be written as below:

$$\frac{\partial C}{\partial t} = -\frac{\partial}{\partial x_i}\left[U_i C\right] + \frac{\partial}{\partial x_i}\left[D \frac{\partial C}{\partial x_i} \right] + S_\alpha \quad (6)$$

Left side of this equation is concentration changes rate in time and left side includes displacement, diffusion and gas pollution source power in environment terms. In pollution emission modeling by air zonal method it is assumed that air in each zone has full mixing and all parameters in that zone are the same. Additionally, we ignored diffusion phenomena and it is assumed that pollution is transferred only through air displacement from one zone to an adjacent one.

Based on above assumptions, mass balance equation for each zone (like zone i) can be written as below [15]:

$$V_i \frac{dC_i}{dt} = \sum_{j \neq i} V_{j-i} C_j - \sum_{j \neq i} V_{i-j} C_j + \sum S \quad (7)$$

In which C_i and V_i are air pollution volume and concentration in zone i respectively, t time and V_{i-j} and V_{j-i} are volume flow rate from zone i to j and from j to i, respectively. C_j is air pollution concentration in zone j and S is pollution source power in zone i. In equation (7), left side indicates pollution change in zone in terms of time. First and second term in the right side indicate input and output pollution to the zone and third term indicates pollution created by pollution source in the zone. Generalizing equation (7) and then simplifying it will give:

$$(1+\frac{\Delta t}{V_i}\sum_{j\neq i}V_{i-j})C_i^{K+1}=(\frac{\Delta t}{V_i}\sum_{j\neq i}V_{j-i})C_j^K+\frac{\Delta t}{V_i}\sum S+C_i^K \tag{8}$$

Considering matrix equation $[A]C_i^{K+1}=[B]$ for above equation, [A] and [B] matrix can be described as equation (9).

$$[A]=(1+\frac{\Delta t}{V_i}\sum_{j\neq i}V_{i-j})$$

$$[B]=(\frac{\Delta t}{V_i}\sum_{j\neq i}V_{j-i})C_j^K+\frac{\Delta t}{V_i}\sum S+C_i^K \tag{9}$$

So after calculating air flow distribution in steady state, pollution distribution can obtained by solving above equation. Please note that for each moment, a system of equation must be solved. For boarders that entering pollution exists, input pollution level will be entered into first term of [B] matrix.

Validation

To make sure of modeling and meshing, a simulation on pollution dispersion from a pollution point-source next to a building has been done. Calculation of field velocity and pollution concentration has been done using air zonal method in mentioned geometry and obtained results are compared with experimental method data and numerical solution. Model geometry is presented in figure 3.

Selected geometry was used in Huber et al's work in 1980 with simulation in a wind tunnel and also in Huber and Selom in 1995 and finally in lee in 1998 with computational fluid dynamic method.

In this study as it is shown in figure 3, a building has been chosen where in it width is 2 times, length 1 time and height of pollution source is 1.2 time of building height. In this study, wind velocity is 2.3 m/s and vertical to Y-Z plane.

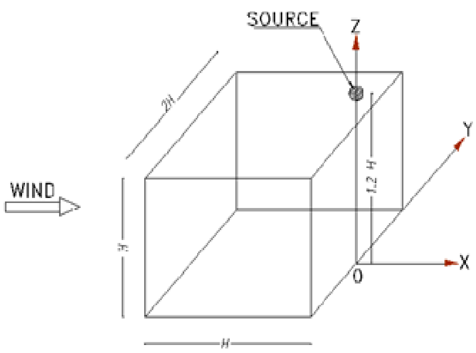

Figure 3. Selected geometry schematic.

Figure 4 shows pollution concentration level in down-stream flow. Experimental results of wind tunnel shows that pollution dispersion level in the wall opposite to wind is negligible and until $X/H=3$ pollution increases and after that until $X/H=7$ it will decreases and after this point remains almost constant. In simulation with computational fluid method, predicting concentration level close to that wall is more than reality and concentration until $X/H=1$ decreases and after that wind tunnel results will take place. Modeling with air zonal method predicts concentration level from the beginning of computational domain and until $X/H=4$ with increasing rate, pollution level increases and until end of the computational domain with mild rate, it will decreases. According to figure 4, predicted concentration level of pollution dispersion with air zonal method increases with higher rate rather than computational fluid dynamic and wind tunnel methods.

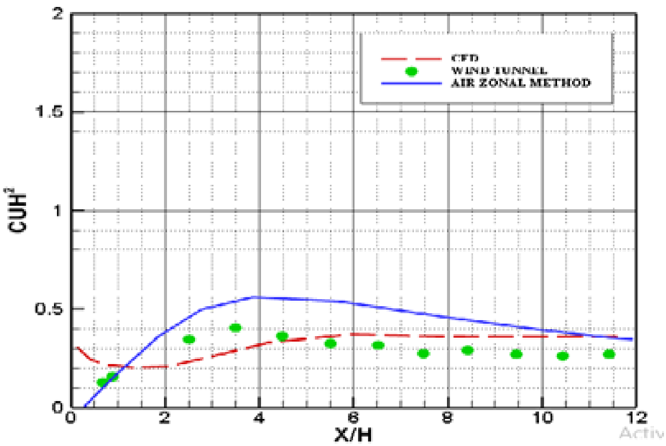

Figure 4. Pollution concentration level in ground diagram.

Figure 5 shows pollution concentration in z axis in a distance equal to X/H=1. Concentration has been investigated from ground surface to 3H height. Results for heights lower than height of pollution source is compatible with wind tunnel data but they are different for higher heights than heights of pollution source in a way that obtained data from CFD modeling have better compatibility rather than air zonal method modeling. As you can see in figure 5, in 1.2H obtained concentration level with air zonal method equals 2 and for wind tunnel is 1.5.

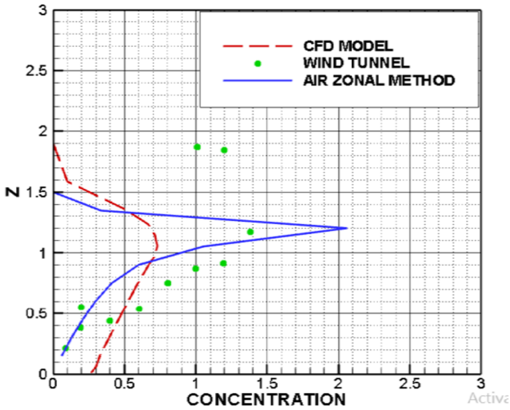

Figure 5. Pollution concentration in Z axis.

Wind velocity effects

After ensuring about obtained results from simulation with air zonal method, now checking wind velocity effect on pollution dispersion on ground will be next step. In this section considering geometry of case study meaning existence of pollution dispersion source in height, one can study pollution dispersion changes on ground for different velocities.

As it is shown in figure 6, when air velocity rises, pollution level on surface decreases and also maximum pollution level transfers in a higher distance from pollution source in a way that when velocity increases from 1 to 2.3 m/s, maximum concentration level on ground transfers from $X/H=2$ to $X/H=4$ and also its level decreases from 0.9 to 0.5. As it is shown in figure 6, in higher velocities, pollution transfers to further zones.

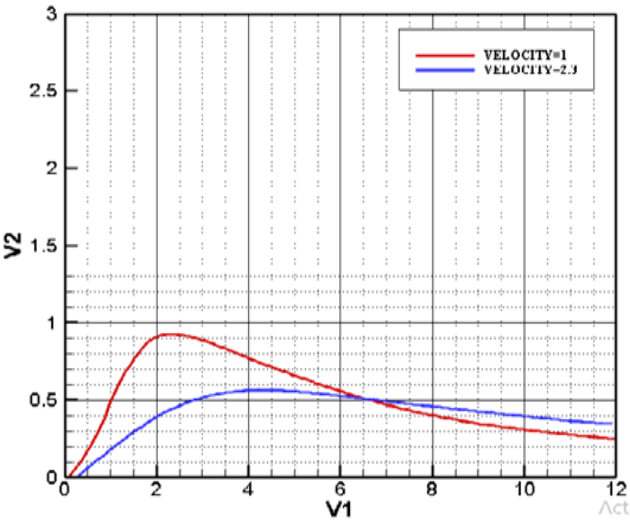

Figure 6. Wind velocity effects on concertation on ground.

Flare height effects

In this section an investigation of the flare height effects on ground surface has been performed. Figure 7 presents pollution level changes on ground surface for different flare heights. As it is shown in figure 6, when height rises, concentration decreases. According to the presented diagram when height increases from 1.2H to 1.5H, maximum pollution level decreases from 0.6 to 0.4. It should be noted that when height increases, pollution start point on ground is created in a further distance from the source.

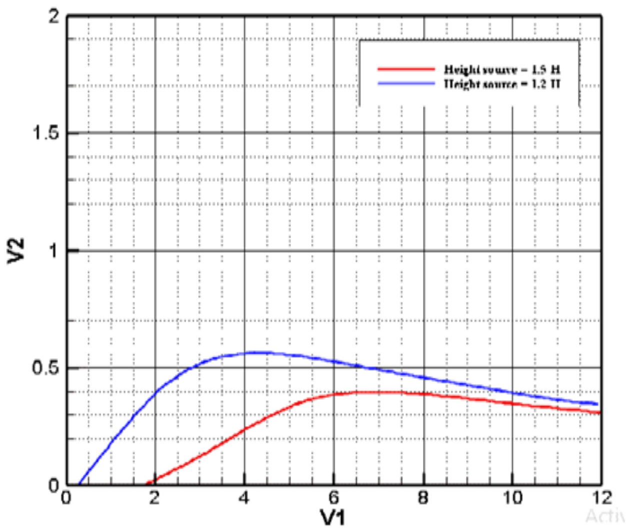

Figure 7. Chimney height effects on concertation distribution on ground.

Barriers effect on pollution dispersion

In this scenario, to investigate barrier effect on pollution dispersion, a building with 1.5H height high is placed on 4H away from pollution source with 1.2H height. Figure 8 shows details of the situation. Wind velocity in this study equals 2.3 m/s and is vertical to Y-Z plane and also concentration levels in the following diagrams are dimensionless.

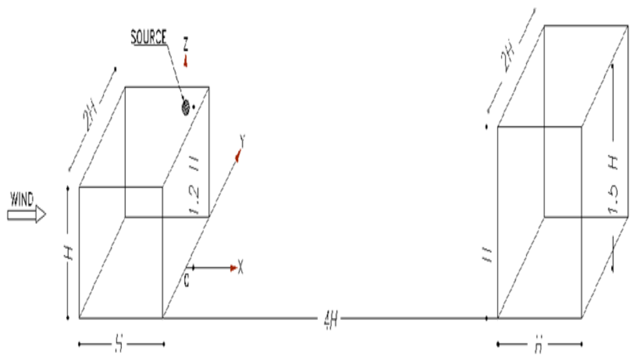

Figure 8. Schematic geometry of case study

Figure 9 shows pollution concentration level on surface ground for two states, one with the wall and the other without the wall in downstream flow. As we can see, pollution level with wall is more than the state without wall and its level is reached to 0.8 from 0.5 and also maximum level is reached in lower distance from source in a way that this maximum level is happened in $X/H=4$ distance for without wall state and for with wall state, maximum pollution is in $X/H=3$.

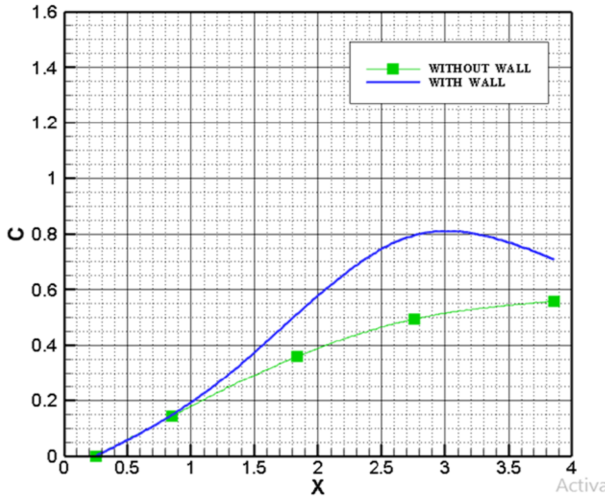

Figure 9. Concentration level diagram on ground surface.

Figure 10 shows pollution concentration in Z axis in a distance equal to X/H=4. As we can see, pollution level at first with barrier is higher than state with no barrier and also its maximum level is on a higher height than the state with no barrier. Also maximum level of pollution concentration with barrier is lower than the state with no barrier.

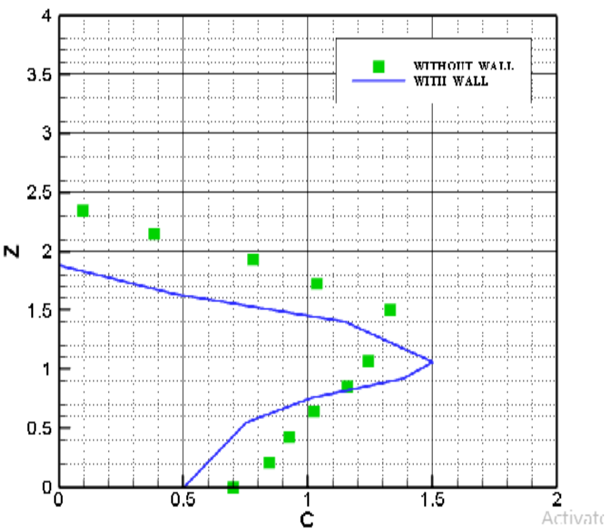

Figure 10. Pollution concentration in Z axis.

Results and discussion

In this paper we wrote a computer program with MATLAB software to develop zonal method for simulation pollution emission in outside environment. In this program using air zonal method, calculation of velocity distribution and mass flow rate of air between zones with zonal method based on pressure has been done. After obtaining velocity in steady state, released pollution and its distribution in each zone has been calculated. And after validating obtained results, we investigated effects of wind velocity and flare height parameters and also pollution dispersion which are listed as below:

- Zonal method is a suitable method to investigate gas pollutant distribution.

- Wind velocity effect on pollution concentration density is in a way that when wind velocity increases from 1 to 2.3 m/s, pollution concentration level on ground surface will decrease from 0.9 to 0.5 and also happens in a further distance from flare.
- With increase in flare height from 1.2H to 1.5H, pollution concentration density on ground surface decreases from 0.6 to 0.5 and also place of pollution start point on ground surface and place of maximum pollution density on ground surface is placed on a further distance from pollution source (flare).
- With addition of barrier in pollution dispersion path, maximum pollution level will increase by 60% and also maximum pollution level will occur in closer distance from source in a way that it has been decreased by 25% from the state with no barrier.

References

[1] Sandberg, H., Lindstorm S., 1987, "A Model for Ventilation by Displacement", *Proceeding of Roomvent*, Stockholm, Sweden.

[2] Voeltzel, A., 1999, "Dynamic Thermal and Airflow Modeling of Large Highly Glazed Spaces", Doctoral Thesis, *University of Lyon*, France.

[3] Inard C., Buty D., "Simulation and Thermal Coupling Between a Radiator and a Room with Zonal Models", *Proceedings of Building Simulation*, France, pp. 113-17., 1991.

[4] Wurtz, E., 1999, "Modelisation Tridimensionnelle des Transfert Thermiques et Aerauliques Dans le Batiment en Environnement Oriennte Objet", *Doctoral Thesis*, *Ecole University*, France.

[5] Bouia, H., 1993, "Modelisation Simplifiee d' Ecoulements de Convection Mixte Interne: Application aux Echanges Thermo- Aerauliques dans les Locaux", *Doctoral Thesis, University of Poitiers*, France.

[6] Chen, Q., Xu, W. "A Zero- Equation Turbulence Model for Indoor Airflow Simulation", *Energy and Buildings*. 1998:28(2):137-44.

[7] Howarth, A.T., 1985, "The Prediction of Air Temperature Variations in Naturally Ventilated Rooms with Convective Heating", *Building Service Engineering Research and Technology*. pp: 6(4):169-75.

[8] Lebran, J., Ngendakumana, P., 1987, "Air Circulation Induced by Heating Emitters and Corresponding Heat Exchanger Along the Wall: Test- Room Results and Modeling". *Proceedings of Roomvent*, Stockholm, Sweden.

[9] Mussy, M., Wurtz, E., Winkelmann, F., Allard, F., 2001, "Generation of a Zonal Model to Simulate Natural Convection in a Room with a Radiative Convective Heater", *Building and Environment*, pp: 36(5):58996.

[10] During, H., 1994, "Consommations Energetiques et Confort Thermique des Locaux Chauffes: Approche par les Modeles Zonaux". *These de Doctorat, INSA de Lyon*, France.

[11] Inard, C., Buty, D., "Simulation and Thermal Coupling Between a Radiator and a Room with Zonal Models", *Proceedings of Building Simulation*. France. pp. 113-17, 1991.

[12] Gschwind, M., Fonzes, G., Loiseau, P.H., Bezian, J.J., Hasebe, T., Fujita, S., Takeda, L., (1995), "A Zonal Model to Simulate a Room Heated by Gas Heat Pump (GHPT, *Proceeding of the 5th International Conference on Air Distribution in Room (Roomvent)*, Hapan. pp: 61-68.

[13] Huang, H., Haghighat, F., "An Integrated Zonal Model for predicting indoor airflow, temperature, and VOC distributions", *ASHRAE transactions* pp:111(1):601-11, 2005.

[14] Daneshgar E., Azimi A., Bahraynian S., 2015, "Developing inverse air zonal method to identify pollution source properties in inside environment", *master's thesis for Shahid chamran ahwaz university*, (in Persian)

[15] De Visscher, A. (2013), "Air Dispersion Modeling. Foundations and Applications*", J.Wiley & Sons, forthcoming.*

[16] Huber, A., Selom F., (1995), wind tunnel simulation review with computational fluid dynamic method, *Energy and Buildings*, pp: 54-55