Evaluation of a Zonal Model for Large Enclosures Using Computational Fluid Dynamics

Jun Gao*1,2, Jia-ning Zhao2, Xiao-dong Li2 and Fu-sheng Gao2

1 Institute of HVAC & Gas Engineering, Tongji University, Shanghai, China
2 School of Municipal & Environmental Engineering, Harbin Institute of Technology, Harbin, China

Abstract
A temperature-based zonal model for large enclosures with combined stratification cooling and natural ventilation, proposed in a previous paper by Gao et al, is evaluated using the CFD (Computational Fluid Dynamics) method in this study. The CFD method adopted in this study using the RNG k-ε model with a differential viscosity is able to represent the non-isotropic thermally stratified flows encountered in the large enclosures. It is used to determine a key parameter, the heat transfer factor between adjacent zones in the zonal model due to temperature difference and turbulent penetration, and also to verify the zonal model for the prediction of thermal stratification in large enclosures. In order to avoid the poor representation of surface heat transfer by empirical/semi-empirical expressions suggested for conventional room enclosures, this study feeds a CFD-based coefficient to the zonal model. This treatment also gains an advantage over the experimental method, which is proved to fail in keeping the similarities of thermal buoyancy and surface convection based on the scale modeling. Furthermore, it responds to the influence of detailed information of local flows on the coefficient. Finally, comparison between the zonal model and CFD is carried out for the enclosure conditioned with the combined stratification cooling and natural ventilation, or single stratification cooling. It is found that the results calculated by the zonal model agree considerably well with those by the CFD.

Keywords: zonal model; stratification cooling; CFD; large enclosure; thermal stratification

1. Introduction
A zonal model has some advantages over the CFD simulation when used for the building energy and environment prediction. On the one hand, a zonal model is inherently very simple to construct and timesaving in calculation. It provides useful information in the macroscopic scale, which neglects the sophisticated phenomena encountered by a microscopic predicting tool, as well as covers up the superfine mathematical language such as the differential equations, discretization scheme, and the viscosity modeling. On the other hand, the CFD method itself is often recognized not easy to utilize, especially for the large enclosures. There are many aspects where the CFD simulation requires to be technically improved and much work still remains to be done for large enclosures. Despite of the extensive problems encountered for the large enclosures when using CFD method, significant advances in these aspects have gradually been made [1-4]. One can believe that all the obstacles will be removed sooner or later and the CFD method will still be the mainstream in case where detailed flow information is needed. The CFD, if well validated and expertly employed, can provide very reliable and reasonable predictions [3].

A zonal model, when extended into unsteady state, has another advantage over a traditional dynamic building energy model, which is always based on the assumption of one-point mode for indoor air temperature throughout the whole space. This assumed pattern has been proved to be unsuitable for many buildings where stably thermal stratifications existed [5,6]. A zonal model discussed in [7] provides a temperature profile due to the thermal stratification, which then produces more accurate building thermal analysis.

This paper and our previous one [7] are not actually intended to present a method to absolutely replace the CFD. This present paper evaluates the temperature-based zonal model using the CFD instead. Based on the thinking of coupled simulation of building energy with the CFD [8-10], the complementarity between them will be expectable in the near future. One significant point is that the consensus among investigators [2,3] holds that the CFD simulation is still an ideal and promising tool. For large enclosures concerned in this study, emphasis is laid on the choice of turbulent model, near-wall treatment method, and the grid-
independent resolution. Eventually, the zonal model is evaluated using the non-isotropic RNG $k-\varepsilon$ turbulence model with a differential viscosity, together with a wall function.

2. A Temperature-based Zonal Model

As discussed in a previous paper [7], for large enclosures conditioned by stratification cooling and natural ventilation, stably thermal stratification is liable to occur except for the occupied zone where the airflow is dominated by the cool jet. In previous study, airflow along high vertical walls induced by the convective heat transfer was considered as one main force driving the pattern of vertical temperature profiles in the space. Heat and air mass transfers through the vertical boundaries in different height were modeled using a wall current model. A precondition for this zonal model is that the whole space should be horizontally divided into a finite number of zones. Based on the air mass and heat balance equations for each settled zone, the zonal model is used to reproduce the thermal stratifications encountered in large enclosures. Impacts of the number of divided zones, the heat transfer factor between adjacent air zones, and the under-relaxation factor for calculations in the model solutions have been discussed. One significant point of a temperature-based zonal model is that it does not introduce any variables related to the air pressure. More detailed information about the model and its calculations can be referred to [7]. This kind of zonal model was firstly investigated by Togari et al [11] for predicting the vertical temperatures in an atrium with a glass wall. They constructed a zonal model taking into consideration some basic flow elements. Extensive experiments were carried out in a scale space to validate their models. However, one problem remained in the application of their model due to the scale method used. The reason lies probably in the fact that similarity criterions for both thermal buoyancy and surface heat convection could not be achieved in the experiments, and that the convection coefficient was assumed almost uniform throughout each whole vertical surface based on an estimate of surface heat balance. Thus, thermal stratifications and building energy in a real large enclosure may be poorly represented. The improved model added such essential flow elements as the multiple cool jets, negative buoyant jet, and natural ventilation into the previous one. When applied to the large spaces, it needs further evaluation to reliably reproduce the thermal stratification occurred. Without reliable experimental data and suitable empirical expressions for the surface convection, the CFD can be used to present the surface coefficient and to further derive a correlation based on the numerical experiment.

3. Justification of the Evaluation Using CFD

The goal of the evaluation of this present investigation is to determine a reasonable range of the value for a heat transfer factor describing heat transfer between adjacent zones in the zonal model due to temperature difference and turbulent penetration, and to value its ability to reproduce the thermal stratifications.

Just as the experimental method of thermal buoyancy phenomenon, theoretical method may also be unable to ensure absolutely correct predictions in any case. However, reliable CFD predictions can most be achieved through exerting a suitable turbulent model, well near-wall treatment, good grid resolution, and reasonable definition of boundary conditions. We can then try to utilize the well-validated and accepted CFD technique to evaluate the zonal model before its application. This method really will not defeat the purpose of evaluation and dependent application of the zonal model because of the advantages of numerical experiment. Section 4 discusses the choice of turbulence model and near-wall treatment for the large enclosures with the low-Reynolds-number thermally stratified flows or sometimes even both high- and low-Reynolds-number flows.

4. Choice of Turbulence Model

The CFD technique is more and more applied to indoor environment design and evaluation. Compared with the experimental method, the CFD reduces greatly the cost and time scale, and avoids the geometric and thermal similarity problems.

Using the CFD to predict indoor airflow and temperature distributions for an enclosure dominated by the thermal stratification is somewhat difficult. An important feature of this kind of flow is that the turbulent viscosity relies significantly on the thermal stratification. Stable stratifications usually dampen turbulence in the vertical direction. Actually, in addition to the turbulent fluctuations suppressed due to the presence of the wall and thermal stratification, much of the space inside a large enclosure is almost stagnant or in the pseudo-laminar state. It has been clarified that the DSM/ASM stress models and MKC/ PDH turbulence models [4,12] can reasonably predict the non-isothermal flow-field with significant non-isotropic effect. However, they all need an increased number of cells and much CPU time, especially for a large enclosure. Since the EVM/EDM-based turbulence model is very simple and popular, it would be very useful and practical if some non-isotropic effect were considered.

The traditional standard $k-\varepsilon$ model [13] is absolutely a high-Reynolds-number model, which was derived by assuming that fully turbulent flow is developed and the effects of molecular viscosity are negligible. It assumes the turbulent viscosity $\mu_t$ outclasses the molecular one $\mu$, and defines it as follows

$$\mu_t = \rho C_p \frac{k^2}{\varepsilon}, \quad C_p = 0.09$$

(1)

Jun Gao
When $k$ and $\varepsilon$ tend to be zero simultaneously in some cases of thermal stratification where little turbulent flow exists, an unstable solution or even divergence can occur. It is an isotropic EVM/EDM model and is not applicable to the thermal stratification and low-Reynolds-number effect for the present need. We applied a non-isotropic EVM/EDM model, RNG $k-\varepsilon$ model with a differential viscosity and heat convection [14,15], to predict the thermally stratified flows encountered. It is derived using a mathematical technique called "Re-Normalized Group" method. A differential equation for turbulent viscosity is obtained as follows to accurately describe the relationship of the effective turbulent viscosity with the effective Reynolds number.

$$
\rho v^2 d\left(\sqrt{Re_e}\right) = 1.72 \frac{\nu d^2}{\sqrt{\nu^2 + 99}}
$$

where $\nu$ is ratio of effective viscosity $\mu_eff$ to molecular viscosity $\mu$, and the effective or turbulence Reynolds number $Re_i$ is defined by

$$
Re_i = \frac{\rho k^2}{\epsilon \mu}
$$

For local low-Reynolds-number flows ($Re<150$), Eq.2 is used to predict the turbulent viscosity, and for high-Reynolds-number flow ($Re>150$), Eq.3 is reduced to Eq.1, but $C_a = 0.085$, which is derived using the RNG theory. This derivation allows the RNG $k-\varepsilon$ model with a differential viscosity to better handle low-Reynolds-number flows. As a non-isotropic turbulence model, its model constants are theoretically derived, and one of them defined for destruction term in the $\varepsilon$ equation is responsive to the effects of rapid strain and streamline curvature. It also provides an analytical formula for turbulent Prandtl numbers rather than using constant values.

In order to reliably represent the surface airflow and heat convection due to the non-equilibrium effect between the turbulence generation and destruction, a non-equilibrium wall function suggested by Kim and Choudhury [16] is used to bridge the viscosity-affected region between the wall and the main flow region. It responds to some pressure-gradient effect caused by thermal stratification and local impingement due to air supply jet along walls. Moreover, a two-layer-based concept adopted by the wall function can effectively budget the turbulence kinetic energy based on the proportions of the viscous sublayer and the assumed fully turbulent layer for the wall-neighboring cells. This wall function therefore relaxes the local equilibrium assumption adopted by the traditional standard wall function proposed by Launder and Spalding [13].

The effect of buoyancy on the generation of $k$, $G_b$, is considered in the thermally stratified flows through Eq.4. While the effect of buoyancy on the generation $\varepsilon$ is relatively not well understood, and is ignored in this study.

$$
G_b = -\beta g \frac{H_i}{\rho \Pr} \frac{\partial T}{\partial y}
$$

where $Pr_i$ is the turbulent Prandtl number for energy, is the coefficient of thermal expansion, and is defined by $-(\partial \rho/\partial T)$. In a separate large enclosures study.

5. Evaluation of the Zonal Model using CFD

This study uses the RNG $k-\varepsilon$ model with a differential viscosity together with the non-equilibrium wall function to evaluate the zonal model constructed for the large enclosures conditioned by stratification cooling and natural ventilation. As discussed in Section 3, the aim of this investigation is to determine the value of the heat transfer factor, $c_i$, defined in the zonal model describing the heat transfer between adjacent vertical zones due to temperature difference and turbulent penetration. This goal is achieved based on the zonal model’s reproducibility of the simulated results of the CFD. First of all, the convection heat transfer coefficient is calculated using the CFD results, instead of the estimate from some empirical or semi-empirical expressions. It means impact of the description of surface convection coefficient in the evaluation of the zonal model can be disregarded.

5.1 Derivation of the value of convection coefficient

The law-of-the-wall for mean temperature in the non-equilibrium wall function is described as the following equations, which remain the same as the standard wall function [13].

$$
T^+ = \begin{cases} 
\Pr y^+, & y^+ \leq y^+_i \\
\Pr \left[ \frac{1}{\kappa} \ln \left( Ey^+ \right) \right], & y^+ > y^+_i
\end{cases}
$$

where $T^+ = (T_a - T_p) \rho c^a_p \kappa^{1/2} / \dot{q}$, $\dot{q}$ is wall heat flux, $T_p$ is temperature at the cell adjacent to wall, $T_a$ is temperature at the wall, $Pr$ is the molecular Prandtl number ($= 0.743$), $Pr_i$ is the energy Prandtl number ($= 0.85$ at the wall), $E$ and $\kappa$ is wall function constant ($= 0.42$ and 9.793 respectively), $y^+_i$ is the nondimensional thermal sublayer thickness and is computed from the intersection of the linear and logarithmic profiles. Approximately, we assume $y^+_i = y^+$. In a separate study which will be published in the near future, we find if the first grid is set at $y^+ = 4.0 \sim 8.0$ for thermally buoyant flows, the RNG $k-\varepsilon$ model with its viscosity equation and the non-equilibrium wall function can ensure close agreement between simulated and experimental results. Therefore, the local heat transfer coefficients predicted by the CFD method can be derived using the linear part of Eq.5 as

$$
\frac{\dot{q}}{(T_a - T_p)} = \frac{\rho c^a_p \kappa^{1/2}}{\Pr y^+}
$$
Substituting Eq.6 into Eq.7, we have

\[ h_c = \frac{\mu c_p}{Pr y_h} \]  

(8)

Based on the energy equilibrium, we can obtain, and then the nominal convective heat transfer coefficient used in the zonal model can be eventually expressed as

\[ h = h_c \frac{T_e - T_p}{T_e - T} \]  

(9)

The mathematical relationship between \( h_c \) and \( h_l \) are described as Fig.1. Using Eq.9, the nominal convective heat transfer coefficient for the zonal model can be obtained by the cell area-weighted average of the grid-based value of \( h_c \) along each surface zone.

\[ h_c = \int_{area} h_c \, dA \]  

Fig.1. Illustration of the Convective Heat Transfer Coefficients in CFD

5.2 Evaluation of the zonal model

First, two cases in a large space of 44m (length) \times 20m (width) \times 16m (high) are carried out to primarily validate the zonal model and determine the range of \( c_b \). (i) Case \( A \), single stratification cooling; (ii) Case \( B \), combined stratification cooling and natural ventilation. In this study, two opposite vertical walls in the direction of length are assumed to be adiabatic and two vertical walls in the direction of width, where air supply nozzles and inflow openings for natural ventilation are mounted, are designed to be symmetrical. Thus, only one vertical wall, half of the floor and the ceiling in a geometrically periodic unit are modeled in the CFD (See Fig.2.). This periodic unit is defined by two opposite periodic air boundaries, within which only one single air supply nozzle, one single inflow and outflow opening are modeled. The grid used for CFD simulation is illustrated in Figs.2. A grid-independent solution is obtained when the space grid size is 0.2m and the first grid near wall is at 0.0065m for the ceiling, 0.00175m for the floor and the wall in the air-conditioned space, and 0.0035 m for the wall in the unconditioned space. The space and boundary zone division used for the zonal model is shown in Fig.3. Zone depth in the unconditioned space is about 0.35 m, which is derived based on the model calculations in the previous paper [7].

Only half of the domain is modeled with a plane of symmetry boundary defined in the geometry (See Fig.2.). Boundary conditions used in the CFD and the zonal model are assembled in Table 1. The interior surface distributions of temperature used are presented in Fig.4. Because radiative heat transfer can be theoretically excluded in the surface balances based on a reapportionment of surface heat flux, it is inessential in the evaluation and is not considered.

\[ \text{Fig.2. Computational Grid for CFD Simulation, 81217 Nodes and 65495 Cells} \]

\[ \text{Fig.3. Zone Division Pattern for Zonal Model Calculation, 32 Vertical Zones} \]

For the CFD simulation, pressure-velocity coupling is achieved by using SIMPLE algorithm [17]. The three-order QUICK differencing scheme [18] is used to discretize the advection terms for Navier-Stokes equations, the energy equations and the turbulent transport equations to reduce the numerical diffusion. Fig.5. shows the averaged convective heat transfer coefficients derived by the CFD for case \( A \) and are fed to the zonal model. Predicted results of vertical temperature profiles are obtained as Fig.6., from which close agreement can be observed. In this case, however, predicted results of zonal model are highly sensitive to the value of heat transfer factor, \( c_b \). The reason is that less air flow in the unconditioned space is observed and heat transfer between adjacent zones due to temperature difference and turbulent penetration is predominant. In Fig.6., a good agreement between the results of zonal model and CFD is achieved by set \( c_b = 18.4 \text{ W/m}^2\text{K} \) for the zones with significant air volume exchange \((\text{ACH}>2.0 /\text{h})\), and \( c_b = 0\sim 2.3 \text{ W/m}^2\text{K} \) for strongly stable stratification exits where air volume exchange is very little \((\text{ACH}<2.0 /\text{h})\).

Fig.7. shows the averaged convective heat transfer coefficients derived by the CFD for case \( B \). Fig.8. shows the temperature profiles predicted by the zonal model for \( c_b = 2.3, 9.2, 18.4, \text{ and } 36.8 \text{ W/m}^2\text{K} \) and their comparisons with the CFD results. Fig.9. presents the predicted ventilation flow rate by the zonal model under different value of \( c_b \) and that by the CFD. From Fig.8., a wide variation of \( c_b \) leads to a slight change in the prediction of zone temperatures. It is also found that the value of \( c_b \) has less effect on the ventilation flow rate. As shown in Fig.9., less than 10% relative difference of ventilation flow rate is observed in the wide range of the value of \( c_b \). Obviously, in the case of the combined system of stratification cooling and natural ventilation where this study lays more...
emphasis, air exchange between zones is highly active, and therefore the prediction of the zonal model is slightly sensitive to the value of \( c_b \).

The following Figs. 10 and 11 present the thermal stratification patterns obtained by the CFD. Air temperature in the lower air conditioned zone is relatively uniform. However, in the upper space it exhibits a significant vertical gradient. Meanwhile, horizontal air temperature is so uniform that the approach of constructing the zonal model in this study is confirmed. To further validate the zonal model, this study extends to carry out some cases based on case B. The height of the inflow openings above the floor is changed in the range of 7 m to 15 m. The boundary conditions are assumed to be unvaried as Table 1.

Table 1. Boundary Conditions used in the CFD Simulations and the Calculation of Zonal Model

| Case name | Nozzle outlets | Return air inlet | Ambient temperature | Ceiling level openings | Low level openings |
|-----------|----------------|------------------|---------------------|-----------------------|-------------------|
| Case A    | 200mmx300mm, \( u_o = 4.75 \text{m/s}, \) | Return air ratio: \( b = 1.00, \) Size: 500x350mm | 26.5 °C | No | No |
| Case B    | 0.6x2x22m², \( C_d = 0.45, \) \( H = 16.0 \text{m} \) | 0.6x2x22m², \( C_d = 0.55, \) \( H = 11.2 \text{m} \) |
to the inherent variation of ventilation flow rate and the averaged temperature in the air-conditioned space due to the change of the height of inflow openings for natural ventilation (See Figs.12 and 13). It is observed that the inflow height at 9.0 m, 1/3 height of the entire upper space, achieves most ventilation flow rate and lowest temperature in the air-conditioned space. The results in Figs.12 and 13 show again a good qualitative agreement between the CFD and zonal model. Some discrepancies are observed in Fig.12. When the inflow openings are near to the cooling jets, zonal model appears to underpredict the ventilation flow rate. This may be caused by the air entrainment of the cooling jets, which theoretically augments the air flow through the openings.

As discussed above, when stratification cooling is combined with natural ventilation in large enclosures, air exchange between zones is very active and the effects of the heat transfer factor $c_b$ on the predicted results of zonal model are not significant. From Figs.12 and 13, the value of 2.3 to 9.2 W/m²K for $c_b$ should be able to sufficiently represent the results and their characteristics of change observed in CFD.

6. Conclusions and Outlook

A turbulence model, RNG $k-\varepsilon$ model with a differential viscosity, together with the non-equilibrium wall function is suggested for thermal stratifications in large enclosures. The CFD method is used to evaluate the zonal model constructed for predicting the temperature profiles. The role of the heat transfer factor, $c_b$, is discussed based on the reproducibility of the CFD results. In the case of combined stratification cooling and natural ventilation, it is found that the predicted results of the zonal model are not sensitive to the value of $c_b$. As a whole, close agreement between

![Fig.10. Temperature Distribution Predicted by CFD, for Case A](image)

![Fig.11. Temperature Distribution Predicted by CFD, for Case B](image)

![Fig.12. Comparison of Ventilation Flow Rate between the Zonal Model and CFD](image)

![Fig.13. Comparison of Air Temperature in the Air-conditioned Space between the Zonal Model and CFD](image)
the CFD and the zonal model for such large enclosures with the combined system is obtained. Using the values of convective heat transfer coefficient from the CFD results avoids the failure of reproducing the surface convection. It seems that this method reduces the independence and integration of the zonal model.

Although the temperature-based zonal model is well validated by the CFD simulation results, it is still unable to be used independently now because the value of $c_v$ are provided by the CFD. It is our future work to develop the zonal model into an independent and integral tool. The heat transfer factor will be physically defined and will be correlated to air laminar and turbulent conduction. This factor and the surface convection coefficient for large enclosures are two crucial parameters for the zonal model to be a practical tool independent from the CFD.

Acknowledgements

This work was supported by Beijing Municipality Key Lab of Heating, Gas Supply, Ventilating and Air Conditioning Engineering.

References

1) Chen, Q., Moser, A. and Huber, A. (1990) Prediction of buoyant, turbulent flow by a low-Reynolds-number $k$-$\varepsilon$ model. ASHRAE Transactions, 96 (1), pp.564-573.

2) Chikamoto, T.S., Murakami, S., and Kato, S. (1992) Numerical simulation of velocity and temperature fields within atrium based on modified $k$-$\varepsilon$ model incorporating damping effect due to thermal stratification. Proceedings of the International Symposium of Room Air Convection and Vent, Japan, pp.501-509.

3) Schild, P.G., Tjelfaat, P.O., and Aiulfi, D. (1995) Guidelines for CFD modeling of atria. ASHRAE Transaction, 101, Part 2, pp.1311-1329

4) Murakami, S., Kato, S., Chikamoto, T.S., et al. (1996) New low-Reynolds-number $k$-$\varepsilon$ model including damping effect due to buoyancy in a stratified flow fields. International Journal of Heat and Mass Transfer, 39 (16), pp.3483-3496.

5) Li, Y. (2000) Buoyancy-driven natural ventilation in a thermally stratified one-zone building. Building and Environment, 35, pp.207-214.

6) Nielson, P.V. (1995) Vertical temperature distribution in a room with displacement ventilation. IEA Annex 26 Report: Energy-efficiency Ventilation of Large Enclosure, Department of Building Technology and Structure Engineering, Aalborg University, Denmark, pp.1-14.

7) Gao, J., Zhao, J. N., Li, X.D., Gao, F.S. (2006) A zonal model for large enclosures with combined stratification cooling and natural ventilation: part 1- model generation and its procedure. Journal of Solar Energy Engineering-Transactions of the ASME, 128, pp.367-375.

8) Negrao, C.O.R. (1998) Integration of computational fluid dynamics with building thermal and mass flow simulation. Energy and Buildings, 27, pp.155-165.

9) Zhao, Z., Chen, Q., Haves, P., et al. (2002) On approaches to couple energy simulation and computational fluid dynamics programs. Building and Environment, 37, pp.857-864.

10) Bartak, M., Beausoleil-Morrison, I., Clarke, J.A. et al. (2002) Integrating of CFD and building simulation. Building and Environment, 37, pp.865-871.

11) Togari, S., Arai, Y., and Miura, K. (1993) A simplified model for predicting vertical temperature distribution in a large space. ASHRAE Transactions, 99 (part 1), pp.84-99.

12) Peng, S.H., Davidson, L. (1999) Computation of turbulent buoyant flows in enclosures with low-Reynolds-number $k$-$\omega$ models. International Journal of Heat and Fluid Flow, 20, pp.172-184.

13) Launder, B.E., Spalding, D.B. (1974) The numerical computation of turbulent flows. Computer Methods in Applied Mechanics and Engineering, 3, pp.269-289.

14) Choudhury, D. (1993) Introduction to the renormalization group method and turbulence modeling. Fluent Inc., Technical Memorandum TM-107.

15) Yakhot, V., Orzag, S.A. (1986) Renormalization group analysis of turbulence: Basic theory. Journal of Scientific Computation, 1, pp.3-11.

16) Kim, S.-E., Choudhury, D. (1995) A near-wall treatment using wall functions sensitized to pressure gradient. ASME FED., 217, Separated and Complex Flows, ASME.

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

18) Leonard, B.P. (1970) A stable and accurate convective modelling procedure based on quadratic upstream interpolation. Computer Methods in Applied Mechanics and Engineering, 19, pp.59-98.