CFD Simulation of Heat Transfer in Sustainable Buildings Envelopes

Haifa El-Sadi
Mechanical Engineering and Technology
Wentworth Institute of Technology
Boston, MA, USA

Abstract. Sustainability of building investment depends upon the technological capability of building such as domes. The objective of this research is to investigate the convective heat transfer in new scenarios of building envelops that attempts to reduce energy consumption and improve occupant's thermal comfort so as to be applied in sustainable buildings. Commercial CFD software package Ansys/Fluent is used in the study and three dimensional steady state model is adopted for the modeling of air flow and heat transfer process. Case studies have been performed for the natural convection in multilayer dome enclosure. The results showed that the heat transfer coefficient increases as the temperature increases as the gap ratio decreases. The interior surface temperature is lower than the exterior surface temperature, which corresponds to the situation of summer climate. The numerical simulation has been validated experimentally.

Keywords: sustainability, CFD, ansys/fluent, heat transfer.

Introduction

Buildings account for about 1/3 of the carbon dioxide emissions and total energy consumption in North America. With the increasing global problem of the greenhouse effect, the technology of sustainable building is now gaining more and more attentions of the building designers. By application of the technology of sustainable buildings, it aims to make the building more energy efficient, to provide the building occupants a comfortable working and living place, and while to reduce its impact on the natural environment.

Energy consumption of the buildings is mainly due to heating, cooling, and lighting, so minimizing this part of energy demand is necessary for the implementation of sustainable buildings. Research is very active in the related areas, such as the optimization of HVAC design, use of renewable energy, and improvement of building envelopes. Among these, the advanced design of building envelopes attracts much attention now since building envelopes are the determining factor of heat loss/gain

of buildings.

Generally, when simulating the heat loss/gain through the building envelope, it is modeled as a multi-layer system. Heat transfer through the envelope is estimated by considering the conduction. Influence of radiation is included for the part of window, and in the determination of the exterior surface temperature. However, in the design of building envelope for the sustainable buildings, air cavities might be integrated.

Influence of convection to the overall thermal performance of the building envelope for sustainable buildings has been studied for the application of the breathing wall (Qiu et al, 2007), the double skin façade (Manz, 2003, Hensen et al, 2002), as well as the air leakage through the wall (Chebil, et al, 2003). However, with the continuing attempts of reducing heat loss and improving indoor environment by improving the design of building envelope, further work on this aspect is still required, since one of the key ideas of these new scenarios is to involve air gaps in the envelope. For example, a multi-layer translucent dome with an air cavity, or a so called "air babble" inflation design, might be adopted in the sports stadium to greatly take advantage of the solar energy. In these design, convective heat transfer might exist due to the temperature gradient between exterior and interior side of the wall, or with the pumping air flow. Investigation of heat transfer property is necessary to study the thermal performance of these new design scenarios in order to evaluate the energy saving ability, and optimum the design.

Extensive theoretical and experimental investigations have been performed on the natural convection in the rectangular cavities, and corrections of heat transfer coefficients have been developed for the engineering implementation. However, concerning the application of dome in the building, study needs to be addressed on the natural convection in nonsquare enclosures. Up to date, investigations have been performed on this. Natural convection heat transfer in a parallelogram-shaped and triangular enclosure was investigated by Asako and Nikmura (1982), and Akinsate and Coleman (1982), respectively. Studies have also been reported on the analysis of natural convection in other shape of enclosures, such as in a trapezoidal enclosure (Moukalled and Acharya ,1997), and in an enclosure having inclined roof (Das and Sahoo, 1998). These results show that the dimension of the enclosure may have the most important influence on the natural convection in these enclosures. Meanwhile, the effect of the shape is also significant. It is pointed out that the corrections for rectangular cavities overestimate the convection heat transfer for the curved cavities (McGowan, et al, 1998). Numerical simulation and experimental investigation have been conducted for natural heat transfer in an isothermal concentric sphere (Grag, 1991, Rathby and Hollands, 1975), which can be regarded as a kind of special dome. A diameter ratio of 2, which is too large to be considered in building dome shape, is adopted in their studies. Laouadi and Atif (2001) presented a comprehensive study of natural convection heat transfer within multi-layer domes. As the isothermal condition of inner and outer surface is assumed, the simulation is simplified to 2D situation. The ratio of gap spacing to the radius of the dome ranges from very small 0.01 to 1, which is fairly large. Corrections were developed for the convection heat transfer as a function of the dome shape and the gap spacing. Their results demonstrate that the convection heat transfer for fully hemisphere domes may reach more than 13% higher than that for low profile domes with small gap spacing, and more than 100% for large gap spacing.

Comparatively, research on forced or mixed convection in a dome is very rare and not many literature was found, though there are plenty of investigations on the forced or mixed convection in a rectangular channel. The studies related to the domed geometry are those on convection in curved rectangular channels. Typical works among these include the research on fully developed angular flow and convection in the channels between two concentric cylinders by Cheng and Akiyama (1970). The flow was assumed to be in the laminar region and uniform heat flux boundary condition was applied in the inner cylinder surface. Targett et al (1995) studied similar flow but taking into account the effect of uniform heat transfer at inner boundary. Investigation by Chillikuri and Humphrey (1981) considered the influence of buoyancy effect. Meanwhile, experimental study has been carried out on this topic (Mori et al, 1977).

In the present work, numerical simulations have been carried out for the natural convection heat transfer in the dome in the novel wall for the green building.

GEOMETRY DESCRIPTION

The case is the natural convection in an enclosed dome. Half of dome was studies due to similarities. The geometry has been revealed in figure 1. Three geometries have been studied with different gap spacing, 0.01 m, 0.02 m, 0.04 m. The gap spacing radius ratio was chosen to be 0.1 to validate the simulation.

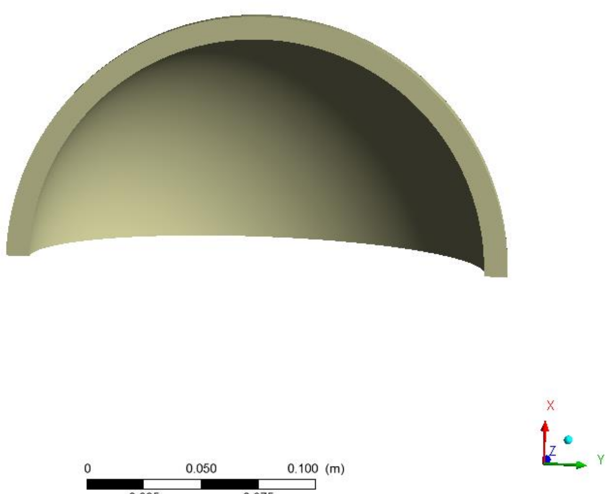


Figure 1. Geometry description-three dimensional half dome

COMPUTATIONAL MODEL

In the present investigation, the dome is modeled by design modeler software. This CFD package uses the finite volume method. It enables the use of different discretization schemes and solution algorithms, together with various types of boundary conditions. As part of the same package, Ansy/design modeler is used to draw the geometry and generate the required grid for the solver. An unstructured grid with triangle is used. The ratio specifies whether the nodes will be denser, and also specifies the intensity of this distribution. Figure 2 shows the mesh topology of the computational domain, the 3-D case generates more than 394094 cells and 74774 nodes. As a convergence criterion in the present work, the solver iterates the equations until it stabilizes at a constant value. The analysis requires 500 iterations for sufficient convergence as shown in Figure 3.

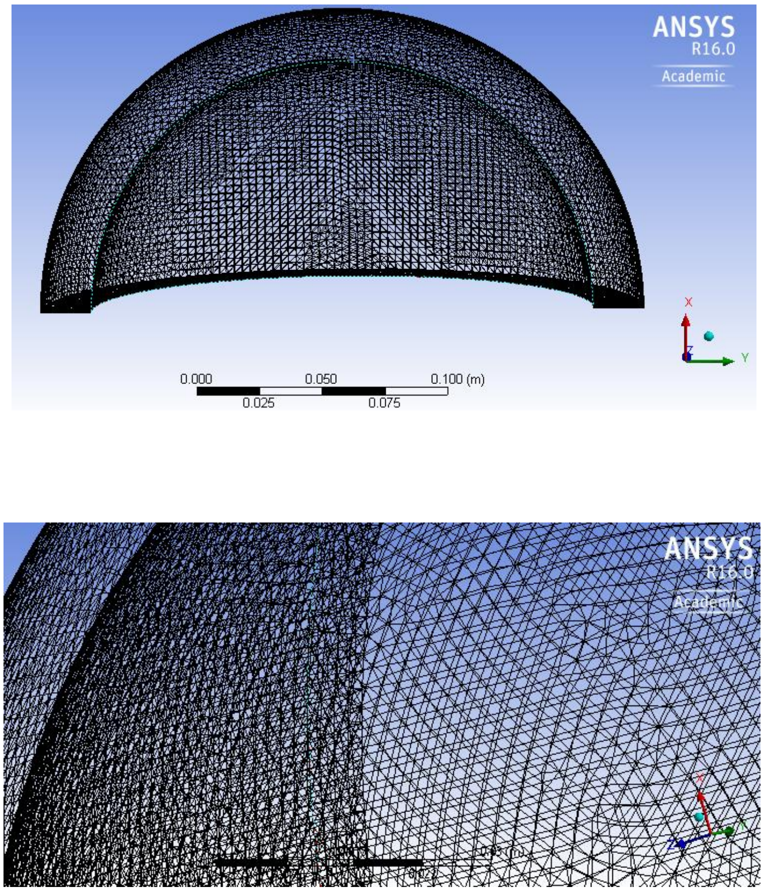


Figure 2. The mesh topology of the computational domain

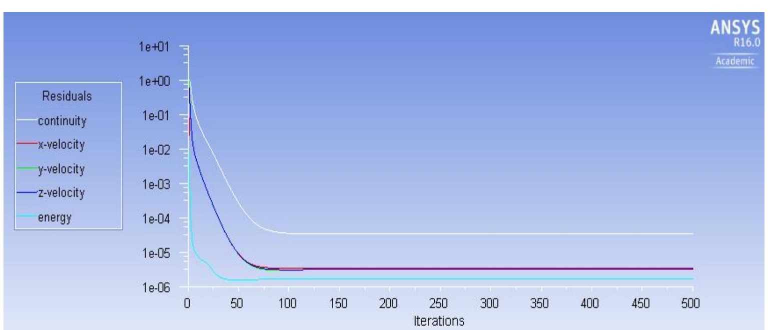


Figure 3. Iterations for sufficient convergence of continuity, momentum and energy equations

Mathematical Equations and Boundary Conditions

The computational fluid dynamic has been used in this work to evaluate the natural ventilation flow performance in three-dimensional dome as shown in figure 1. For three-dimensional incompressible flows, the momentum and energy conservation equations are:

$$\frac{\partial u}{\partial x}+\frac{\partial \upsilon}{\partial \mathrm{y}}+\frac{\partial w}{\partial z}=0 \qquad \textbf{(1)}$$

$$\nabla\,.\,(\rho\vec{\upsilon}\vec{\upsilon})=-\nabla p+\nabla.(\overline{\overline{\tau}})+\rho\vec{g} \qquad \textbf{(2)}$$

$$\nabla\,.\,(\rho\vec{u}T)=-\frac{1}{C_p}\nabla.(q) \qquad \textbf{(3)}$$

The commercial CFD package FLUENT is used for the simulation in this study. According to the geometry characteristic of the building envelope, 3D simulation is adopted. Flow is assumed to be in steady state. The air flow is considered as incompressible and the Boussinesq approximation is used to represent the variation of the density.

$$\rho=\rho_0(1-\beta\,\Delta T) \qquad (4)$$

Where ρ is the density of the flow and β is the thermal expansion coefficient, this model is used if the temperature difference in the domain is small.
For the case of natural convection, the flow region is determined by the Rayleigh number (Ra). If Ra is less than 10^7, then the flow is also in the laminar region. For the design of an air gap less than 0.1m, the Ra number is below 10^7, even under the condition of indoor-exterior temperature gradient being 500C. Upon these considerations, the airflow in the cavity is modeled as laminar in this study.
Upon the simulation, the average Nusselt number (Nu) at the exterior and interior layer is calculated to represent the heat transfer in the air cavity, and is defined as:

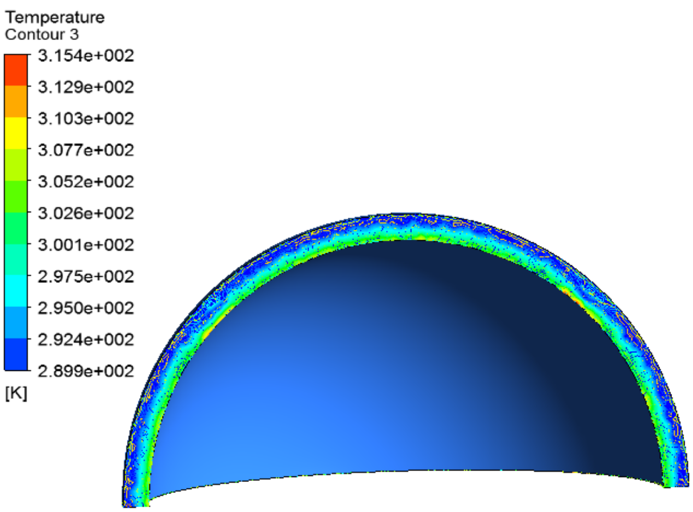
$$Nu=\frac{q_i}{q_{cond}}=\frac{q_o}{q_{cond}}$$

Where qi is the heat flux at the interior surface (W)
q_o is the heat flux at the exterior surface (W)
q_i cond is the pure conduction heat flux (W)

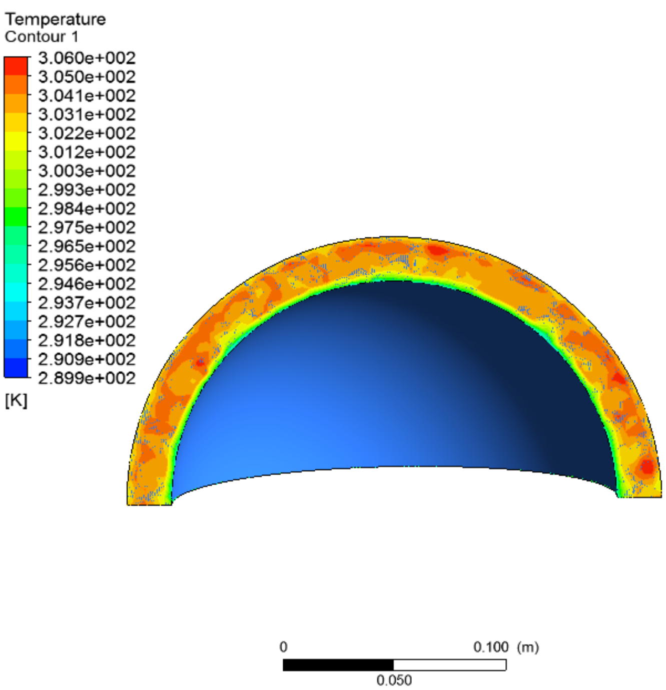
The building envelope in the engineering implementation is usually a multi-layer structure, with different heat transfer process in each layer. The model in the FLUENT is adopted with conduction considered for the solid layer, while air flow and convection model for the cavity or indoor environment.

Numerical simulation has been carried out for the natural convection heat transfer in the dome to set up a basic model for the domed envelope. Simulations have been conducted with the variation of following parameters:

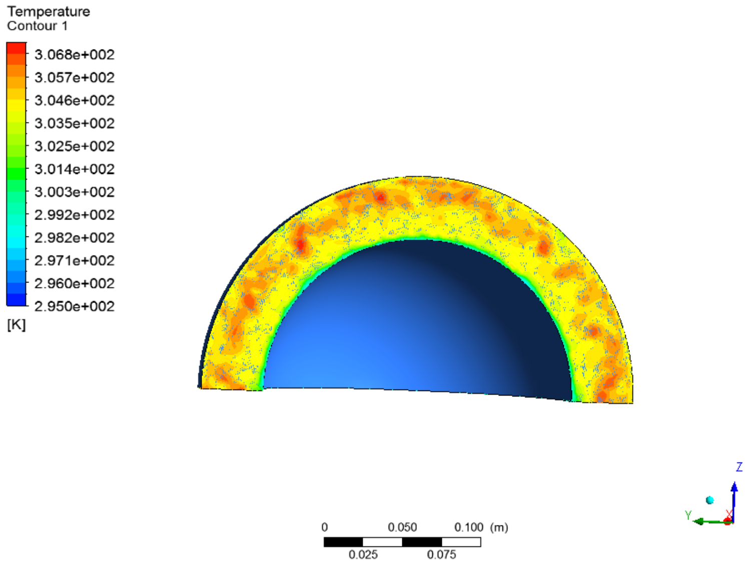
- Gap spacing-radius ratio
- Rayleigh number



(a)



(b)



(c)

Figure 4. the contour of the air with different with different gap spacing, (a) 0.01 m, (b) 0.02 m, (c) 0.04 m

Figure 4 shows the contour of temperature at different gap spacing. The interior temperature decreasing with increasing the gap spacing. In the future more work will be considered by choosing thermal mass material to replace the glass. The interior surface temperature is lower than the exterior surface temperature, which corresponds to the situation of summer climate. Natural convection can be observed mostly occurs at the lower part of the dome, while temperature stratification can be found at the top part of the dome. For natural convection, the air flow in a concentric sphere is first simulated for the validation of the simulation method, this is mainly due to the availability of experimental and numerical simulation in the literature. To correspond with the situation in the literature, a gap spacing-radius of 1.0 is adopted, and simulation is carried out for different Rayleigh numbers.

Validation

Table 1 shows the *Nu* obtained, and compares with the numerical results and data calculated by using empirical equation, which was based on the summarization of the experimental data. It can be seen that the current results agree well with the results by Laouadi and Atif (2001). There is about 10% error between numerical simulation and empirical equation calculated data. This is mainly because of the error in the summarization of the experimental data, as about 10% error was mentioned in the literature (Raithby and Holland, 1975). The below comparison shows that the current simulation work could provide satisfactory results.

Table 1 *Nu* for concentric sphere

Ra	Current simulation	Laouadi and Atif (2001)	Empirical equation (Raithby and Holland, 1975)
21000	2.58	2.345	2.0511
42000	2.94	2.760	2.4392
91000	3.047	3.283	2.9594

The gap spacing-radius ratio is 0.1. Figures 3 and 4 illustrate the temperature profile and stream function contour obtained. A natural convection flow can be seen from these figure 5. For *Ra* of 0.91×10^5, the *Nu* calculated is 3.047, about the same results with that obtained by Laouadi and Atif (2001), which is 3.283.

CONCLUSION

A CFD simulation study has been carried out for the heat transfer property of the advanced building envelope that integrates the air flow in its cavity, for their potential applications in the sustainable buildings. Case studies include the natural convection in multi-layer dome enclosure. The simulation results have been validated by comparing with the experimental data in the literature. The interior surface temperature is lower than the exterior surface temperature, which corresponds to the situation of summer climate. Suggestions have been presented concerning the optimum design in the engineering implementation.

NOMENCLATURE

u velocity component in x - direction

υ velocity component in y - direction

ω velocity component in z - direction

ρ density

p pressure

τ shear stress

g acceleration due to gravity

T temperature

C_p specific heat

q heat transfer

ρ_0 density of the flow

β thermal expansion cofficient

Ra Rayleigh number

Nu Nusslet number

ACKNOWLEDGMENTS

I would like to thank Professor Jackson for his support

References

[1] Akinsate, V. A., Coleman, J. A. Heat transfer by steady state free convection in triangular enclosures. International Journal of Heat and Mass Transfer, Vol. 25, 1982, p 991-998

[2] Asako, Y., Nikamura H. Heat transfer in a parallelogram shaped enclosure. Bulletin of JSME, Vol. 25, 1982, p 1419-1427

[3] Chebil, S., Galanis, N., Zmeureanu, R. 2003. Computer simulation of thermal impact of air infiltration through multilayered exterior walls. Eighth International IBPSA Conference, Eindhoven, Netherlands

[4] Cheng, K. C., Akiyama, M. Laminar forced convection heat transfer in curved rectangular channels. International Journal of Heat and Mass Transfer, Vol. 13, 1979, p 471-490

[5] Chilikuri, R., Humphery, J. A. C. Numerical computation of buoyancy induced recirculation in curved square duct laminar flow. International Journal of Heat and Mass Transfer, Vol. 24, 1981, p 305-314

[6] Das, S. Sahoo R. Velocity-pressure formulation for convective flow inside enclosure with top quadratic inclined roof. Journal of Energy of Heat and Mass Transfer, Vol. 20, 1998, p 55-64

[7] Grag, V. G. Natural convection between concentric spheres. International Journal of Heat and Mass Transfer, Vol. 35, 1991, p 1935-1945

[8] Hensen, J., Bartak, M., Drkal, F. 2002. Modeling and simulation of double-skin façade systems. ASHEA Transactions, Vol. 108, part2, p1251-1258

[9] Laouadi, A., Atif, M. R. Natural convection heat transfer within multi-layer domes. International Journal of Heat and Mass Transfer, Vol. 44, 2001, p 1973-1981

[10] Manz, H. 2003. Numerical simulation of heat transfer by natural convection in cavities of façade elements. Energy and Buildings, Vol. 35, n3, p305-311

[11] McGowan, A. G., Desjarlais, A. O., Wright, J.L. Simulation and testing of pyramid and barrel vault skylights. ASHRAE Transactions: Symposia, 1998, p 832-844

[12] Mori, Y., Uchida, Y., Ikon, T. Forced convection heat transfer in a curved channel with a square cross section. International Journal of Heat and Mass Transfer, Vol. 14, 1971, p 1787-1805

[13] Moulkalled, F., Acharya, S. Buoyancy-induced heat transfer in partially divided trapezoidal cavities. Numerical Heat Transfer, Part A, Vol. 32, 1997, p 787-810

[14] Qiu, K., Haghighat, F., Guarracino, G., Thermal behavior of diffusive building envelope: State-of-the art view. Journal of Advances on Building Energy Research, No 1, 2007

[15] Raithby, G. D., Hollands, K. G. T. A general method of obtaining approximate solutions to laminar and turbulent free convection problems. Advances in Heat Transfer, Vol. 11, 1975, p 266-315

[16] Targett, M. J., Retallich, W. B., Churchill, S. W. Local coefficients for forced convection in curved rectangular channels of large aspect ratio with unequal heating. International Journal of Heat and Mass Transfer, Vol. 38, 1995, p 1989-1999

[17] Yin, S. H., Powe, R. E., Scanlan, J. A., Bishop, E. H. Natural convection flow patterns in sphere annuli. International Journal of Heat and Mass Transfer, Vol. 16, 1973, p 1785-1795

[18] I. S. Jacobs and C. P. Bean, "Fine particles, thin films and exchange anisotropy," in Magnetism, vol. III, G. T. Rado and H. Suhl, Eds. New York: Academic, 1963, pp. 271–350.