Characteristics of Heat Transfer of Buoyancy-driven Rectangular Cavity with Various Aspect Ratios

JUHEE LEE
Department of ICT and automotive engineering, Hoseo University, KOREA
juheelee@hoseo.edu http://www.hoseo.ac.kr
YOUNJUN LEE
BEL technology, KOREA
leeyj@beltec.co.kr

Abstract: Energy saving is one of the global issues in these days. To improve insulation performance for energy saving, multiple layers of the window glasses and walls are employed in architectural engineering. In designing energy-saving buildings, the heat transfer through a large windows or wall becomes critical. The buoyancy-driven air flow between window glasses or wall layers is a key of the heat transfer in the building. A numerical code for the heat transfer problem is developed considering gravity and buoyancy-driven momentum at Ra=1e5. A temperature-based energy equation is combined with the incompressible Navier-Stokes equation by using Boussinesq approximation. A finite volume method based on a fully-implicit second-order is used to discretize and solve the momentum equation on unstructured grids composed of arbitrary-shaped cells. The integrations of the governing equation are discretized in the finite volume manner using a collocated arrangement of variables. A SIMPLE algorithm is employed to solve the pressure and momentum equations for the incompressible flow instead of a decoupled continuity equation. The numerical methods are verified and the reliability of the method is assessed. The characteristics of heat transfer and flow in cavity with various aspect ratios are closely investigated with the newly developed code.

Key-Words: - Natural convection, Aspect ratio, Computational methods, Heat transfer, Cavity

1 Introduction

Heat transfer by the natural convection is of interest in many industrial and architectural applications. There are a variety of applications, such as thermal insulation analysis, cooling of electric equipment, fluid flow in cavity between large windows, convective heat flow in a gap in window frames and so on. The industrial and building applications show the geometrical complexity of calculation domains and therefore, the use of unstructured meshes has become general. The main reason is the ability of unstructured meshes to discretize arbitrary and complex domains and the ease of local and adaptive grid refinement which enhances the efficiency of the solution as well as solution accuracy. In addition, solution algorithms for computing flows on unstructured grids have been continuously developed. Among the discretization methods, the finite volume methods (FVM) are most widely used for computational fluid dynamic (CFD) applications. This is mainly due to the inherent conservativeness of FVM and ease of understanding. These FVMs are capable to accommodate arbitrary polyhedral grids composed of cells of various topologies.

A single numerical code is not fully satisfied the all of the requirements in industrial purpose and thus, a lot of CFD program have been developed. Developing a CFD code based on the authors' previous study [1, 2] to calculate the natural convection phenomena next to the window cavity and window frame, we have been implementing a code based on the SIMPLE method [3] for solving two- and three-dimensional incompressible flow using unstructured grids. In this study, flow and temperature according to various aspect ratio of 2d cavity will be carefully investigated, after validation of the temperature field in an incompressible flow with a rectangular cavity.

2 Numerical Methods

In the derivation of the governing equations of fluid dynamics, the Eulerian methods, spatially fixed control volume, is employed instead of Lagrangian methods. The laminar flow of an incompressible Newtonian fluid is assumed.

2.1 Finite volume methods

The equations for the transport of a scalar variable can be written in the following form:

$$\frac{\partial}{\partial t}\int_V \rho\phi dV + \int_S \left[\rho\phi\vec{v} - \Gamma_\phi \vec{\nabla}\phi\right]\cdot d\vec{S} = \int_V Q_{\phi,V} dV + \int_S \vec{Q}_{\phi,S}\cdot d\vec{S} \qquad (1)$$

where ϕ, Γ_{ϕ}, $Q_{\phi,S}$, and $Q_{\phi,V}$ stand for the transported variable, the diffusion coefficient, the surface exchange terms and volume sources, respectively. The momentum and energy equations can also be written in the form of (1) except an additional diffusion terms in the momentum equation.

The conservation equation of the continuity equation, for a control volume drive from the (1) implies that the rate of change of the mass inside the control volume   is equal to the difference between inflow and outflow mass fluxes across the volume surface. In integral form, the continuity equation can be written as follows:

$$\frac{\partial}{\partial t}\int_V \rho dV + \int_S [\rho\vec{v}]\cdot d\vec{S} = 0 \qquad (2)$$

where, ρ is the fluid density and   is the velocity. In incompressible flow, the first term in (2) is zero and the convection term cannot be ignored. More details including discretization, pressure correction and SIMPLE methods can be found in Ref. 2.

2.2 Validation

Third order diffusion term is chosen to reduce the checkerboard pressure oscillation problem generally encountered in a collocated method for solving incompressible flow. 2-dimensional cavity flow with natural convection has been a test and validation problem for the viscous flow solver developed in this study. The geometry is simple and easy to handle. However, its nonlinear nature is the characteristics of natural convection, and provides a good test of the computational procedure. The configuration used in this study is shown in Fig. 1. The high and low temperatures are located at each side and the others are adiabatic. The Ra is 1×10^5 and the properties of the medium are listed in Table 1.

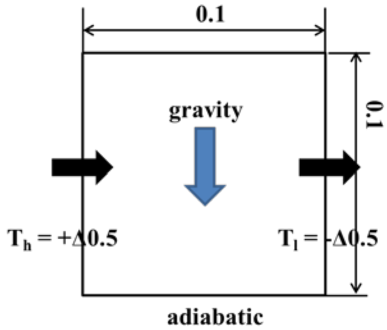


Fig. 1 computational domain and boundary conditions

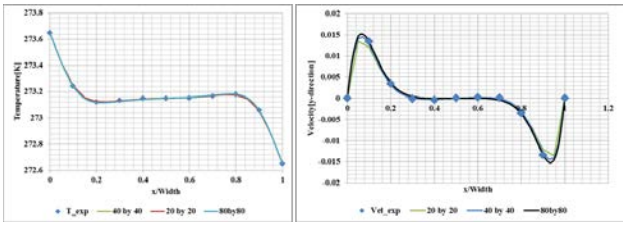


Fig. 2 Comparison of temperature and velocity at y/H=0.5

Fig. 3 Stream line and temperature contours

Table 1 properties of fluid

Name	Value	Name	Value
Cp	9.7703e2	ρ	1.2
K	2.51e-2	Pr	0.71
G	9.81	Ra	1.028e5
β	3.41e-3	μ	1.82e-5
α	2.14e5		

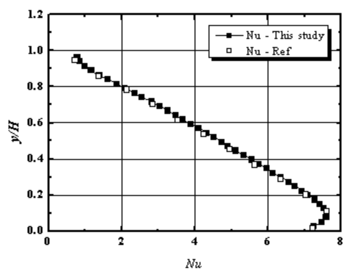


Fig. 4 Comparison of local Nu

A good agreement with existing numerical data [4] is obtained as shown in Fig. 2. The computation was performed according to consecutive numbers of meshes (20x20, 40x40, and 80x80). The flow stream and temperature contour are shown in Fig. 3. There are two currents cause inner circulation circuits. The fluid obtains heat energy from the hot wall in left side and loses it to the cold wall in right side. There is no transfer of heat through the adiabatic horizontal walls. It is expected that boundary layers each hot and cold wall are exited. According to the distance between two walls, two boundary layers interfere with each other and the interference between two currents causes change of the flow pattern and heat transfer significantly.

In Fig. 4, the local Nu along the high-temperature wall obtained in this study, shows good agreement with numerical result [3]. The values of local Nu are highest next to the adiabatic bottom wall and are decreasing linearly along the wall. The local Nu is calculated as follows:

$$Nu = \frac{d\overline{T}}{d\overline{x}} = \frac{d(T_w - T_{fluid})/(T_{max} - T_{min})}{dx/W} \qquad (3)$$

The local Nu linearly dependents temperature difference between wall and cell next to it. The reason for the highest Nu is that the cooled fluid from the opposite cold wall is directly brought into the hot wall and the temperature differences are large. In this area, the heat transfer by convection and conduction are the same direction.

3 Results

The purpose of this study is the numerical analysis of natural convection in the various-sized cavity of the window and its frame with computational fluid dynamics developed by the authors.

The cavity is inevitably built according to the design manufacturing purposes and there are a diversity of the cavity sizes and shapes in a single window. The unintended heat transfer by natural convection through the cavity is occurred and the prevention of this heat transfer is important for improving the insulation performance of window. The numerical analysis is performed and the physics of the heat transfer through the cavity is carefully analyzed according to AR (aspect ratios) from 0.5 to 4. Two different types of AR are considered (height and width) and Fig. 5 shows the shapes with respect to AR.

In left side of Fig. 5, the height is fixed and the width is varied according to AR from 0.5 to 4. On the other hand, the fixed width cases are shown in right side of Fig. 5. For convenience, the letter ‘y’ is labeled at the end of the AR of the fixed width. Interestingly, two cases with AR=0.5 vs. AR=2.0y and AR=2.0 vs. AR=0.5y have geometrical similarity and comparison of characteristics of fluid and heat transfer phenomena between them will be performed.

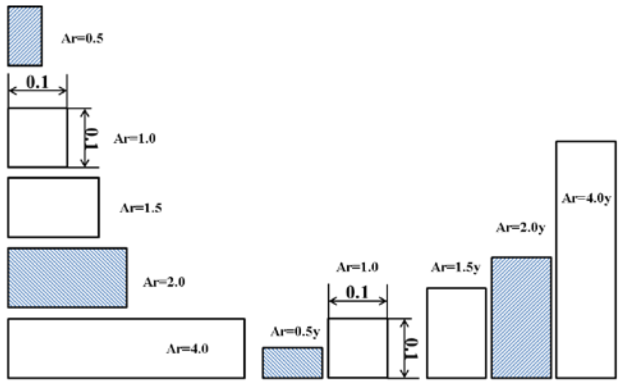


Fig. 5 Schematic view of cavity with different ARs

The heat flux density with respect to AR is shown in Fig. 6. For convenience, the AR_width and AR_height are labeled for fixed height and fixed width respectively. In terms of AR_width, the heat flux densities are decreased according to the increasing AR linearly. Along the high temperature wall, the heat transfer occurs by heat conduction due to temperature difference between wall and fluid and the fluid obtained the energy is accelerated in the opposite direction of the gravity by buoyancy effect. The buoyancy-driven flow transfers the energy to the other side of the cavity. During this process, the friction because of the viscosity on the wall is attended. Along the distance between two horizontal walls, the energy loss by the fluid friction increases and the velocity in the boundary layer decreases inevitably. This induces the decrease in temperature gradient in boundary layer and heat transfer according to AR.

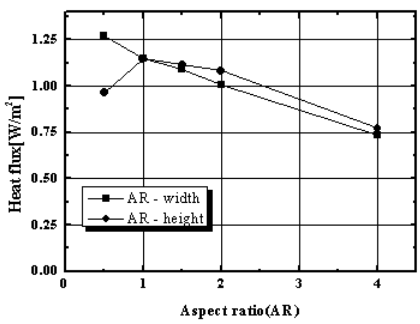
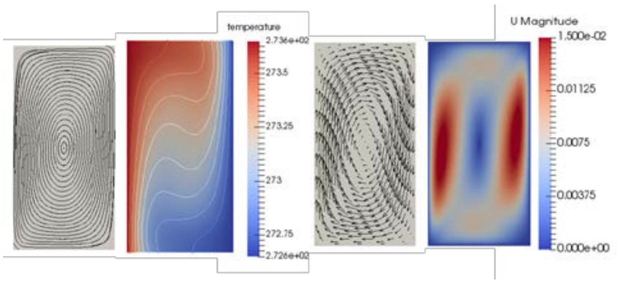
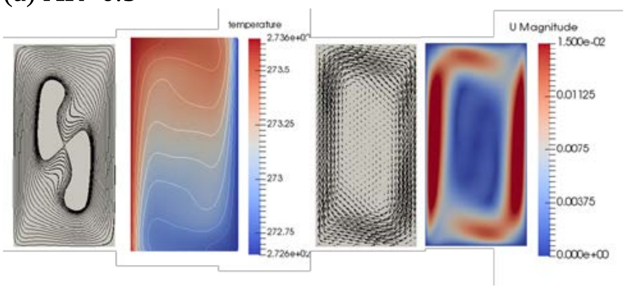


Fig. 6 Comparison of heat flow density

The thickness of the vertical plate is 0.0278 at y=H of Ra=1x10^5 [5]. In this study, the thickness is thicker than that of vertical plate because the thickness of the boundary layer is not zero at the lower-left corner and the thickness of the boundary layer increases along the wall distance. In case of AR=0.5, the width of the cavity is as close as 0.05 and there is a mutual interference of the boundary layers between two vertical walls. Two boundary layers are merged at each horizontal walls and number of recirculation area is reduced to one from two of AR=1.0. These reduce the wall friction and increase the heat flow density as shown in Fig. 7.



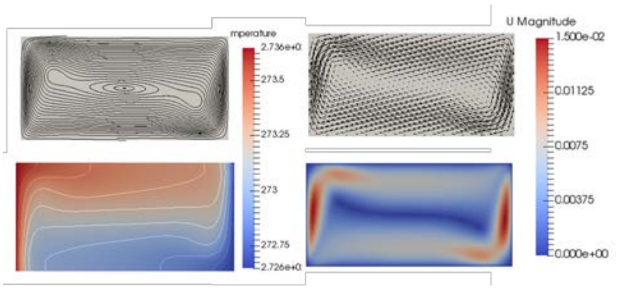
(a) AR=0.5



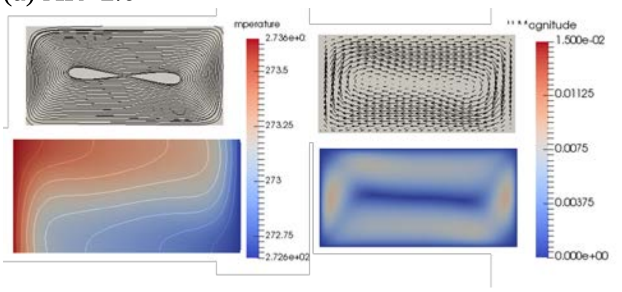
(b) AR=2.0y

Fig. 7 Stream line, temperature, velocity vector and velocity magnitude.

The two cases with AR=2.0 and 0.5y have geometrical similarity as shown in Fig. 8. Actual height of the AR=2.0 is two times long as that of the AR=0.5y. For the AR=0.5y, the buoyancy is small and acceleration along the hot wall is insufficient. From the fact, it deduces that the momentum and velocity in the boundary layer are smaller than that of the AR=2.0. Before the development of the boundary layer sufficiently, the flow meets the vertical wall and thus, there is no interference between two vertical walls. There is two re-circulation zones with nearly horizontal in Fig. 8 whereas the two re-circulation zones are stretched from left-top to right-bottom for AR=2.0.



(a) AR=2.0



(b) AR=0.5y

Fig. 8 Stream line, temperature, velocity vector and velocity magnitude.

4 Conclusion

In this study, the numerical algorithms for incompressible Navier-Stokes flow and energy equation on unstructured grids with arbitrary shaped cells have been developed. Third order diffusion method is employed to avoid the checkerboard pressure oscillation generally encountered in a collocated method for solving incompressible flow. The temperature equation is used as an energy equation. The numerical algorithm used to solve the final linear equations is derived from the SIMPLE algorithm. The numerical methods including energy equation used have been validated and a good agreement with existing reference data and numerical data is obtained.

We confirmed that the geometry of two-dimensional rectangular cavity with Ra=1x10^5 is quite simple, but flow and heat transfer is largely varied according to aspect ratio. For AR=0.5, there is a boundary layer interference between both vertical walls and only single re-circulation is observed. The flow velocity in the boundary layer is smaller than that of AR=2.0y but the heat flux density is large. The AR=0.5y and 2.0 have a geometrical similarity. There is no flow interference between both boundaries because of the smallest hot wall length, the velocity of the AR=0.5y are small. The smallest heat flux density is shown among four cases of various AR. In the future, the detail analyses of velocity and heat flux according the ARs are investigated.

Acknowledgement
This work was supported by the Korea Institute of Energy Technology Evaluation and Planning (KETEP) and the Ministry of Trade, Industry & Energy (MOTIE) of the Republic of Korea (No. 20152000000210).

References:

[1] Lee, Juhee, Lee, Youngjun, Flow and heat transfer of high aspect-ratio rectangular driven by natural convection, *Seoul International Conference on Engineering and Applied Sciences*, 2017.

[2] Lee, Juhee, Lee, Youngjun, Finite Volume Method for Flow Prediction using Unstructured Meshes, *WASET, London*, 2015

[3] Patankar, S. V., Numerical Heat Transfer and Fluid Flow, Hemisphere Publishing Corp, New York, 1980.

[4] D. C. Wan, B. S. V. Patnaik, and G. W. Wei, 2001, A NEW BENCHMARK QUALITY SOLUTION FOR THE BUOYANCY-DRIVEN CAVITY BY DISCRETE SINGULAR CONVOLUTION, *Numerical Heat Transfer, Part B*, Vol. 40, pp199-228.

[5] Frank Kreith, Raj M. Manglik, Mark S. Bohn, 2011, *Principles of heat transfer 7th Edition*, Cengage Learning, Chap 4