An Investigation of Transitional Phenomena from Laminar to Turbulent Natural Convection using Compressible Direct Numerical Simulation

ChungGang Li & Makoto Tsubokura
Advanced Institute for Computational Science, RIKEN, Kobe, Japan

Abstract The transitional phenomena from laminar to turbulent natural convection and the development process inside the channel are investigated using compressible direct numerical simulation (DNS). Numerical method of Roe scheme with preconditioning and dual time stepping are used for addressing natural convection flows with large temperature differences, which are low speed but the densities are variable. The results are qualitatively well consistent with the experimental data [1] and the transition point can be accurately captured. In addition, the development process respected to time can be clearly identified for four stages, which are laminar, unstable process, reliminarization and turbulence. After reaching the quasi-steady state, it can be observed that the laminar, transition and turbulence coexist in the same flow filed. Most important of all, the transitional phenomena are naturally induced by the effects of interactions between the buoyancy and shear stress without adding any fluctuations at inlet. It means that the numerical scheme and physical model adopted in this study has the potential to be a universal case for estimating the accuracy of turbulence model because the characteristics of parameters-free and independence from inlet condition.

INTRODUCTION

The transition from laminar to turbulent natural convection in an open-ended finite length channel is one of the most important subjects for heat transfer applications. Experimentally, Miyamoto et al. [1] is a pioneer about this topic. However, there are three major challenges to simulate this kind of flow, which are the availability of using compressible program on low speed regions, the appropriate boundary conditions at inlet and outlet, and the accuracy of capturing turbulent motions in small scales.

Because of the inapplicability of Boussinesq approximation under the high temperature difference, adopting the compressible program at very low Mach numbers is necessary. Weiss and Smith [2] proposed a preconditioning matrix matching with Roe scheme and dual time stepping to simulate a natural convection in a two dimensional concentric circles.

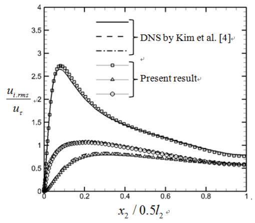
For the boundary conditions, in incompressible flows, generally, methods of matching a mass conservation or based on Bernoulli's equation are used. However, the above methods are not suited to the compressible flows. Fu et al. [3] combined the Non-reflection and absorbing boundary conditions and extended them to low Mach numbers to be suited to the problem of the channel flow with natural convection.

To compare with the numerical turbulent solvers of the Reynolds-averaged Navier-Stokes equations (RANS) and LES, DNS has the merit of the reliability.

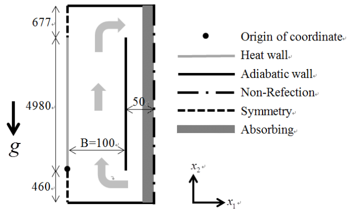
The aim of this study is thus to investigate the transitional phenomena from laminar to turbulent natural convection emplying compressible DNS. The statistics data is compared with Miyamoto et al. [1] to observe the realibility of the numerical scheme.The development process respected to time is also investigated to clearly idenfy different stages for the turbulent natural convection according to the flow field. Finally, it can be observed that the laminar, transition and turbulence coexist in the same flow filed. It means that the present numerical scheme and physical model has the potential to be a universal case for estimating the accuracy of turbulence model because the turbulence is generated automatically without any modification on inlet and the numerical scheme is toward to parameters-free.

PHYSICAL MODEL AND NUMEIRCAL METHOD

The fully developed channel flow adopting the same resolution with Kim et. al [4] is first used to validate the program and the result shown in Fig. 1(a) is in excellent agreement with Kim et. al [4]. With the same resolution, the target physical model is a vertically placed channel flow with constant heat flux $104W/m^2$ on the wall accrding to [1] shown in Fig. 1(b). The total resolution is $4000 \times 192 \times 150$ and the simulation has been run on K-computer in RIKEN AICS. In the right hand side of the channel, an extra computational domain and absorbing zone are added. The governing equation is a complete Navier-Stoke equation for compressible flow. Owing to extremely low Mach numbers, the Roe scheme with a preconditioning method is then adopted. For the boundary conditions, at the inlet, the flowrate can't be obtained beforehand and at the outlet, the flow should leave the computational domain without too much reflection to pollute the flow field. Therefore, the hybrid boundary condition at low Mach numbers [3] which combines non-reflection and absorbing boundary conditions is adopted.



(a) Turbulence intensities

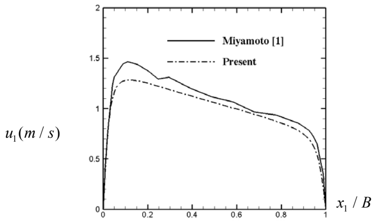


(b) Physical model (the unit of the size is mm)

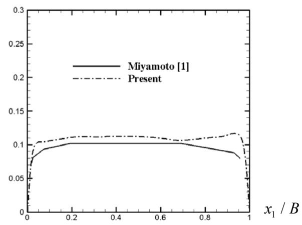
Figure 1.

RESULTS AND CONCLUSION

Fig. 2(a) and Fig. 2(b) show the mean velocity profile and turbulence intensities in x_1 direction at $x_2 = 3.89m$. The present result is qualitatively well consistent and shows the same trend with [1].



(a) Mean velocity profile



(b) turbulence intensity

Figure 2. Statistical quantities

Fig. 3 shows the thermal contour for seven different time steps. According to the flow field, there are four stages, which are laminar ($t = 8.0s$), unstable stage ($t = 11.2s$), relaminarization and ($t = 12.2s$) and turbulence ($t = 17.6s$), can be clearly identified.

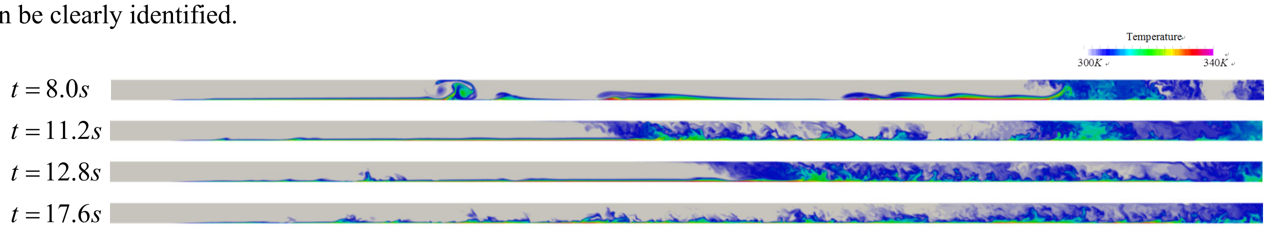
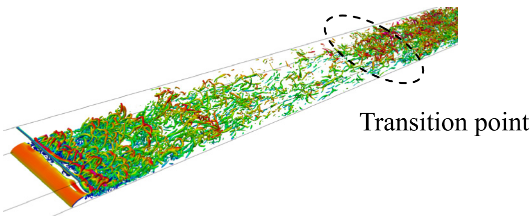
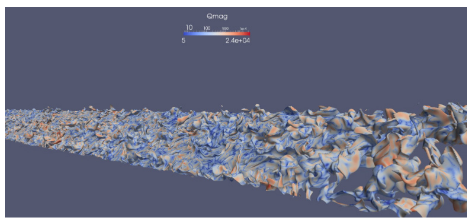


Figure 3. The variation of thermal contour

Fig. 4 shows the turbulence structures at the statistically steady state. In Fig. 4(a), the hairpins can be obviously obesrved near the inlet and the transition point can be accurately captured, which shows the numerical scheme is very accurate. Fig. 4(b) shows how turbulence structures with higher temperature is formed and split up into smaller fragments.



(a) The iso-surface of Q criterion



(b) The isothermal surface of $T = 310K$ contoured by Q

Figure 4. The turbulence structures

References

[1] M. Miyamoto ,Y. Katoh, J. Kurima, H. Sasaki. *Proceedings of the eighth international heat transfer conference* 1593-1598, 1986.
[2] M. Weiss, A. Smith. *AIAA* **33**: 2050-2056, 1995.
[3] WS. Fu, WH. Wang and CG. Li. *International Journal of Thermal Sciences* **84** : 48-61, 2014.
[4] Kim J, Moin P and Moser R. J. Fluid Mech. **177 :** 133-166, 1987.