COMPARISON OF TWO K-ϵ MODELS FOR SIMULATION OF TURBULENT NATURAL CONVECTION IN A SQUARE CAVITY

C. BEGHEIN, C. INARD, F. ALLARD
Centre de Thermique de l'INSA de Lyon, URA CNRS 1372
INSA, bât. 307, 20 av. A. Einstein,
69621 Villeurbanne, France.

ABSTRACT

Turbulent natural convection in a square cavity filled with air and submitted to horizontal temperature gradients is studied numerically. Turbulence is modelled using k-ϵ models, the resulting differential equations are solved with SIMPLER algorithm. Two distinct k-ϵ models are compared. The former is derived from the well known "high Reynolds number k-ϵ model": the constants of this model are identical to ones usually used; nevertheless, the boundary conditions are not expressed on the first internal node out of the viscous boundary layer, but at the wall. The latter is a "low Reynolds number model" based on a previous work of Abrous et al.. The low Reynolds number effects are modelled using the turbulent viscosity concept defined as a function of the local turbulent Reynolds number, and the mesh includes the viscous sublayers.

KEYWORDS Turbulence, k-ϵ Model, Convection

INTRODUCTION

The aim of this study is to compare the behaviors of two different k-ϵ models, for case of convection in a thermally driven square cavity. A first series of results obtained with the reference standard k-ϵ model proposed by Prof. C.J. Hoogendoorn and dr. R.A.W.M. Henkes enables us to validate our code. This model is then compared to the low-Reynolds number k-ϵ model proposed by Abrous et al..

PHYSICAL AND MATHEMATICAL MODELS

Physical model

The physical model to be studied here is a square cavity filled with air whose upper and lower horizontal walls are adiabatic, vertical walls are submitted to constant temperature levels. The fluid motion is induced by the temperature difference between the left hot wall and the right cold one.

Mathematical model

The turbulent behavior of fluid is modelled via the eddy viscosity concept proposed by Boussinesq which relates the turbulent stresses $-u'_i u'_j$ to the mean velocity gradients. The turbulent heat fluxes $-u'_j T'$ are expressed from Reynolds analogy between momentum and heat. The turbulent viscosity is calculated in each point of the cavity from the two-equation k-ϵ model of turbulence. The resulting equations written in their dimensionless form are the following:

Continuity

$$\frac{\partial \overline{u_i}}{\partial x_i}=0 \tag{1}$$

Momentum

$$\begin{aligned}\overline{u_j}\frac{\partial \overline{u_i}}{\partial x_j}&=-\frac{\partial \overline{P}}{\partial x_i}+\frac{1}{Gr^{1/2}}\frac{\partial}{\partial x_j}\left[(1+\nu_t)\frac{\partial \overline{u_i}}{\partial x_j}\right]\\&+\frac{1}{Gr^{1/2}}\frac{\partial}{\partial x_j}\left[(1+\nu_t)\frac{\partial \overline{u_j}}{\partial x_i}\right]+\delta_{i2}g\beta(\overline{T}-0.5)-\frac{2}{3}\frac{\partial k}{\partial x_i}\end{aligned} \tag{2}$$

Energy

$$\overline{u_j}\frac{\partial \overline{T}}{\partial x_j}=\frac{1}{Gr^{1/2}}\frac{\partial}{\partial x_j}\left[\left(\frac{1}{Pr}+\frac{\nu_t}{\sigma_T}\right)\frac{\partial \overline{T}}{\partial x_j}\right] \tag{3}$$

Turbulent kinetic energy

$$\begin{aligned}\overline{u_j}\frac{\partial k}{\partial x_j}&=\frac{1}{Gr^{1/2}}\frac{\partial}{\partial x_j}\left[\left(1+\frac{\nu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]\\&+\frac{1}{Gr^{1/2}}(P_k+G_k)-\epsilon\end{aligned} \tag{4}$$

Dissipation rate of turbulent kinetic energy

$$\begin{aligned}\overline{u_j}\frac{\partial \epsilon}{\partial x_j}&=\frac{1}{Gr^{1/2}}\frac{\partial}{\partial x_i}\left[\left(1+\frac{\nu_t}{\sigma_\epsilon}\right)\frac{\partial \epsilon}{\partial x_i}\right]\\&+\frac{1}{Gr^{1/2}}\left(C_{\epsilon1}f_1(P_k+C_{\epsilon3}G_k)\right)\frac{\epsilon}{k}-C_{\epsilon2}f_2\frac{\epsilon^2}{k}\end{aligned} \tag{5}$$

Turbulent viscosity

$$\nu_t=Gr^{1/2}C_\mu f_\mu\frac{k^2}{\epsilon} \tag{6}$$

With:

$$P_k = \frac{1}{Gr^{1/2}} \frac{\nu_t}{2} \left(\frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i} \right)^2 \tag{7}$$

$$G_k = -\frac{1}{Gr^{1/2}} \frac{\nu_t}{\sigma_T} \frac{\partial \overline{T}}{\partial x_j} \delta_{j2} \tag{8}$$

The non-dimensional variables are:

$$\begin{aligned} &x_i = x^*_i/H;\ u_i = u^*_i/u_0 \\ &T = (T^* - T_C)/(T_H - T_C);\ P = P^*/(\rho u_0)^2 \\ &k = k^*/u_0^2;\ \epsilon = \epsilon^*/(u_0^3/H);\ \nu_t = \nu_t^*/\nu \end{aligned} \tag{9}$$

As the buoyancy process is much stronger than the diffusion process for turbulent flows, the velocities are made dimensionless with the buoyant velocity u_0:

$$u_0 = (g\beta \Delta TH)^{1/2} \tag{10}$$

In this study, 2 k-ϵ models are investigated:

1. High Reynolds number k-ϵ model proposed by Henkes and Hoogendoorn in Eurotherm-Ercoftac Workshop (Henkes 1989)
2. Low Reynolds number k-ϵ model developed by Abrous et al. (1984)

In the standard k-ϵ model, no wall functions are used in order to obtain grid independent results for the turbulent quantities; indeed, the dissipation rate of turbulent kinetic energy is set infinite at the wall. $C_{\epsilon1}f_1$, $C_{\epsilon2}f_2$, $C_\mu f_\mu$ are constant within the whole flow domain. These constants, the turbulent Prandtl numbers and the boundary condition for ϵ are the following:

$$\begin{aligned} &C_{\epsilon1}f_1 = 1.44;\ C_{\epsilon2}f_2 = 1.92;\ C_{\epsilon3} = \tanh\left|\frac{\overline{v}}{\overline{u}}\right|;\ C_\mu f_\mu = .09 \\ &\sigma_T = 0.9;\ \sigma_k = 1.0;\ \sigma_\epsilon = 1.3 \\ &\epsilon_{wall} = \infty \end{aligned} \tag{11}$$

Low Reynolds modelling consists in using damping functions f_1, f_2, f_μ, rather than constants, to account for the laminar behavior of fluid close to the walls. Hence, the first discretization node is located within the viscous sublayer.

In the model developed by Abrous et al., the values of $C_{\epsilon1}f_1$, $C_{\epsilon2}f_2$, $C_\mu f_\mu$, σ_T, σ_k, σ_ϵ are as recommended by Launder and Spalding (1974):

$$\begin{aligned} &C_{\epsilon1}f_1 = 1.44,\ C_{\epsilon2}f_2 = 1.92 \\ &C_\mu f_\mu = .09\exp[-3.4/(1+R_t/50)^2] \quad where\ R_t = Gr^{1/2}k^2/\epsilon \\ &\sigma_T = 1.0,\ \sigma_k = 1.0,\ \sigma_\epsilon = 1.3 \end{aligned} \tag{12}$$

In order to take into account the viscous effects in the viscous sublayer, f_μ depends on the local turbulence Reynolds number R_t. $C_{\epsilon 3}$, which is .4 near horizontal walls and 1.44 near vertical walls, is assigned the following value:

$$C_{\epsilon 3}=.7+(1.44-.7)\frac{|\overline{u}|}{\sqrt{\overline{u^2}+\overline{v^2}}} \tag{13}$$

The boundary condition for ϵ is deduced from the expression of the balance of k in the viscous sublayer (To and Humphrey 1986):

$$\epsilon_{wall}=\frac{2}{Gr^{1/2}}\left(\frac{\partial k^{1/2}}{\partial x}\right)^2_{wall} \tag{14}$$

For these two models, the turbulent kinetic energy is zero at the wall.

NUMERICAL METHOD

The equations to be solved are advection-diffusion equations, coupled with pressure, velocities, temperature and turbulent quantities. They are spatially discretized over a staggered grid by the finite difference method and then integrated over control volumes. The system of equations is implicitly solved by the SIMPLER (Semi Implicit Method for Pressure Linked Equations Revised) iterative process proposed by Patankar (1980). The equations are discretized with the Power-Law scheme. The solution option of the finite difference equations is the line by line Tri-Diagonal Matrix Algorithm (TDMA) (Anderson et al. 1983). Special attention is paid to the implicit under-relaxation of the set of equations, the explicit under-relaxation of the turbulent quantities, and to the k and ϵ equations source terms linearization. Convergence of the SIMPLER algorithm is reached when the residuals of all the equations are very small.

RESULTS & DISCUSSION

Qualitative results obtained with the standard k-ϵ model are presented. Simulations were carried out, for a Rayleigh number range between 10^8 and 10^{12}. Streamlines and isopleths of temperature and turbulent viscosity are plotted in figures 1,2,3. In the core of the cavity, a stratification of temperature pattern is to be noticed. The size of this area increases with Rayleigh number, while the thermal boundary layer thickness decreases (figures 2.a, 2.b, 2.c). So does the dynamic boundary layer thickness; the hydraulic jump which occurs for low Rayleigh numbers (figure 1.a) disappears for higher Rayleigh numbers (figures 1.b, 1.c): the higher the Rayleigh number, the higher the turbulent viscosity, the stronger the diffusion process. The behavior of the fluid is turbulent in the top left and bottom right parts of the cavity, these areas strech along the horizontal walls, when Rayleigh number is increased (figures 3.a, 3.b, 3.c).

The values of the average Nusselt number at hot wall and the maximum vertical velocity at half the cavity height, obtained with the two k-ϵ models, and for different Rayleigh numbers, were computed. These results highlight the different behaviors of the standard k-ϵ model and the low Reynolds number k-ϵ model proposed by Abrous et al.. From Figure 4, it is seen that the transition region between the laminar and the turbulent behaviors differs. The transition occurs at $Ra=10^9$ for the standard model and at $Ra=10^{10}$ for Abrous low Reynolds number model. The slopes of the curves in the turbulent regime are not the same; the standard model overpredicts the average heat transfer rate at hot wall. Maximum values of the non dimensional vertical velocity at mid height of the cavity are plotted in Figure 5. Although the two models predict identical values for the laminar regime ($Ra=10^8$), a great discrepancy is to be noticed for the maximum vertical velocity at the laminar-turbulent transition. In the turbulent regime, the values of non dimensional vertical velocity decrease and get closer for both models.

CONCLUSIONS

Turbulent natural convection in a thermally driven square cavity was numerically investigated. A standard k-ϵ model with boundary conditions for turbulent quantities imposed at the wall and a low Reynolds number k-ϵ model were used. The numerical procedure is the SIMPLER algorithm. Discrepancies between the two models are to be noticed, for prediction of significant quantities such as the average Nusselt number at hot wall, or the maximum vertical velocity at half the cavity height. Moreover, the laminar-turbulent transition region, for different values of Rayleigh numbers, is not the same for both models. Other results by direct simulations or experiments need therefore to be performed, in order to predict the turbulent motion with a more accurate turbulence model.

REFERENCES

A. ABROUS, A.F. EMERY, F. KAZEMZADEH, "Turbulent free convection in rooms in presence of drafts, cold windows and solar radiation", Heat transfer in enclosures, ASME winter annual meeting, New Orleans, Louisiana, HTD vol.39, pp. 49-54 (1984).

D.A. ANDERSON, J.C. TANNEHILL, R.H. PLETCHER, "Computational fluid mechanics and heat transfer", Hemisphere, Washington (1983).

R.A.W.M. HENKES, "Natural convection boundary layers", PhD thesis, Delft University of Technology, Faculty of Applied Physics, The Netherlands (1989).

B.E. LAUNDER, D.B. SPALDING, "Numerical computation of turbulent flows", Computer Methods in Applied Mechanics and Engineering, vol.3, pp. 269-289, North-Holland (1974).

S.V. PATANKAR, "Numerical heat transfer and fluid flow", Mac Graw Hill, London (1980).

W.M. TO, J.A.C. HUMPHREY, "Numerical simulation of buoyant, turbulent flow - 1. Free convection along a heated, vertical, flat plate", Int. J. Heat Mass Transfer, vol.29, no 4, pp. 573-592 (1986).

ACKNOWLEDGEMENTS

Calculations were performed on the IBM 3090 supercomputer of the CNUSC (Centre National Universitaire Sud de Calcul) of Montpellier (France), within the frame of the C3NI (Centre de Compétence en Calcul Numérique Intensif).