BBAA VI International Colloquium on:
Bluff Bodies Aerodynamics & Applications
Milano, Italy, July, 20-24 2008

PLUME BUOYANCY EFFECTS IN POLLUTION DISPERSION AROUND A CUBE

Jalal Abedi and Mohammad Omidyeganeh
Department of Chemical and Petroleum Engineering, University of Calgary, Calgary, Alberta, Canada, T2N1N4 Phone: (403) 220-5594
e-mail:jabedi@ucalgary.ca

EXTENDED ABSTRACT

Atmospheric stability has significant effects on air flow and dispersion around buildings in urban area. Unstable conditions cause more turbulent flow and rapid dispersion of pollution. Simulating the instability conditions accurately, mostly modeled by buoyancy effects, would be crucial for reliable concentration reports. Air pollution disposed near a building often have higher than the allowable concentration of various contaminants. When the pollutants are past the acceptable levels the impact is adverse. It necessitates some action to ensure the safety those who may come into contact with the pollutants. If the levels are high enough urgent action must be taken which often includes the evacuation of the premises.

In the present work, at the beginning, simpler buoyancy driven flow in a square cavity has been investigated to utilize the code with buoyancy considerations and difficulties in application. A finite volume code has been developed for two-dimensional flow in a cavity with two sided walls kept in constant hot and cold temperatures. The top and bottom walls are adiabatic which inhibit the heat loss from the cavity. The main goal of this sample problem is to simulate the air flow affected by buoyancy from stable initial condition and reach to a stable steady state condition. Therefore the laminar flow was investigated by solving Navier-Stokes equations and coupled continuity and energy equations.

$$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=-\frac{\partial p}{\partial x}+\Pr\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right) \tag{1}$$

$$u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}=-\frac{\partial p}{\partial y}+\Pr\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right)+Ra\Pr\left(T-T_0\right) \tag{2}$$

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \tag{3}$$

$$u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}=\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2} \tag{4}$$

A fully implicit method with QUICK scheme for convection terms and Central Difference scheme for diffusion terms is used for Navier-Stokes equations' discretisation. The energy equation is implemented the Hybrid scheme for convection terms. In addition to discretisation method SIMPLER algorithm was applied into the code to solve the velocity-pressure coupling difficulties.

Initial condition set zero for all variables and no-slip boundary condition is used for velocity. The results have been obtained for several Rayleigh numbers to investigate the effects of the parameters on the flow inside the cavity. The Prandtl number kept constant for all cases assuming the fluid inside the cavity does not change and the thermal parameters are not a strong function of temperature.

$$Ra = \frac{\rho^2 \; g \; \Delta T \; L^3 \; \Pr}{T_0 \; \mu^2} \qquad (5)$$

$$\Pr = \frac{\mu \; c_p}{k} \qquad (6)$$

In the above equations, ρ is the density of air, ΔT is the temperature difference between cold and hot wall, L is the dimension of the cavity, μ is the viscosity of air, T_0 is the cold wall temperature, c_p is the specific heat of air, and k is the thermal conductivity of air.

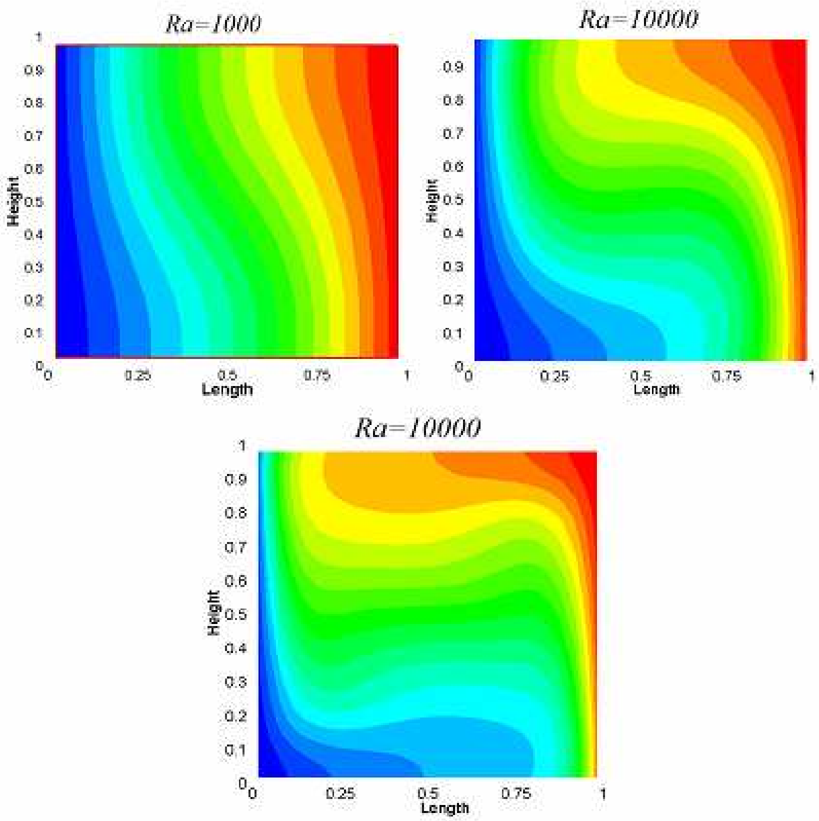

Figure 1: Temperature distribution inside the cavity for different Rayleigh numbers

Figure 1 shows that At *Ra=1000*, isotherms are parallel to the heated walls, indicating that most of the heat transfer is by heat conduction. The effect of convection is seen as the departure of the isotherms from the vertical for higher Rayleigh numbers. As the Ra increases, the effect of convection is more pronounced in the isotherms. Temperature gradients are more severe near the vertical walls, but diminish in the center.

The vortex is generated by the horizontal temperature gradient across the section. This gradient is positive everywhere, giving rise to negative (counter clockwise rotation) vorticity at low Rayleigh number. The velocity vectors are shown in figure 2. At higher Rayleigh number, Heat transfer by convection in the viscous boundary layers alters the temperature distribution to such an extent that temperature gradients in the center are close to zero, or change sign, thus promoting negative Vorticity causes the development of secondary vortices in the core.

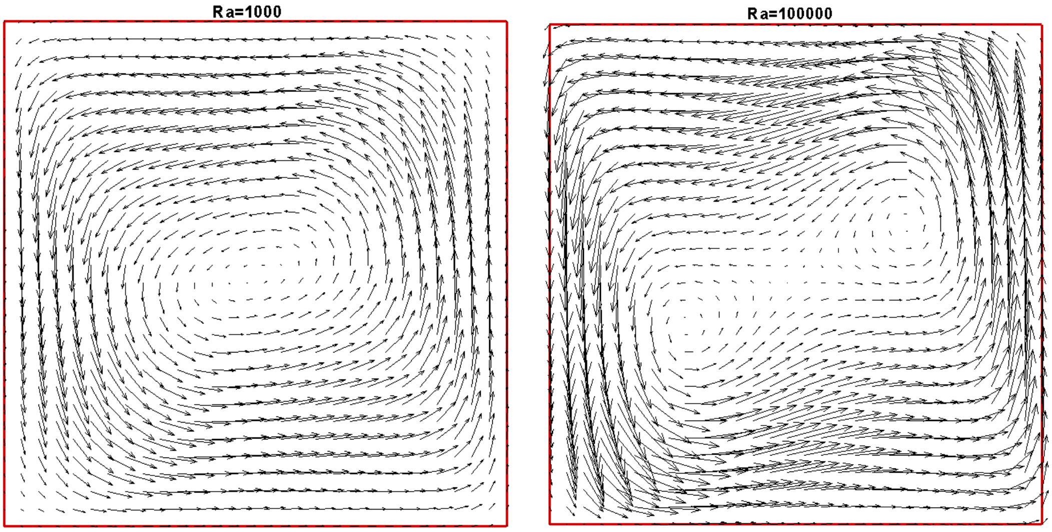

Figure 2: Velocity vectors at two different Rayleigh numbers

The most important characteristic of the flow is the rate of heat transfer across cavity which is increased with Rayleigh number as shown in figure 3.

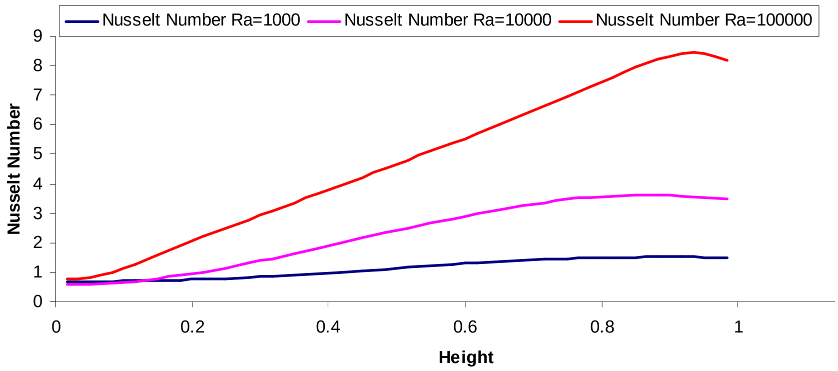

Figure 3: Nusselt number across the hot wall

Next step is to add energy equation into the proposed code which have developed based on finite volume method to simulate air flow at the atmospheric boundary layer to predict pollution dispersion from a variety of sources. The flow field around a single building placed in a surface boundary layer is fully turbulent and very complex. There is separation and recirculation on each surface of the building and when this problem is extended to an arbitrary configuration of a building understanding the conditions of pollutant dispersion is clearly compounded. To address this work will be to apply the solution concept of Large Eddy Simulation (LES) to turbulent flow over and around buildings with sharp edges and corners. Due to the large-scale unsteady air motions around bluff bodies, LES can be expected to be more accurate and reliable than the RANS models which are commonly used to calculate mean flow and the effects of turbulence on mean flow properties.

In this step the purpose of our work is to apply the solution concept of LES to turbulent flow over and around buildings with sharp edges and corners. An implicit finite volume method is used to solve the incompressible Navier-Stokes equations on staggered grids. Convection terms in momentum equations are approximated by TVD (Total Variation Diminishing) with UMIST limiter function. In this study a central difference scheme is used for diffusion terms and Adams-Bashforth scheme for time marching. The PISO algorithm (Pressure Implicit with Splitting of Operators) makes up the main body of the code. It involves two corrector steps which enable the robust convergence behavior in addition to less computational effort than conventional SIMPLE, SIMPLER or SIMPLEC algorithms. The finite volume solver uses the top-hat filter. A Smagorinsky-Lilly SGS model was applied in LES which assumes that the smallest turbulent eddies are almost isotropic.

The code consists of two separate parts. First part is to solve Navier-Stokes equations to get to the statistically steady state condition. The second part facilitates our ability to solve the mass equation in addition to the energy equation and Navier-Stokes equations to find the pollution dispersion. The effect of the buoyancy force on the dispersion of contaminants is investigated and compared with the case where this force had been neglected.

A 10 cm cube representing a model building is mounted on a 1.2 x 2.4 m flat plate. The model is mounted with the front edge 2.3 cm downstream from the leading edge of the plate. The contaminant source with diameter of 0.09 H_b (H_b is the building height) is located behind on the ground level centerline at a downwind distance of 0.25 H_b from the base of the building. This problem has been studied to validate the numerical results with experimental data, reported by Zhang et al. (used TEMPEST code to carry out numerical studies of this particular problem). The numerical simulations are reported correspond to the case with different Froude numbers. The computed concentration profile is compared with the experimental data downstream of the building. For the comparison, the normalized concentration is defined as $x = cU_b H_b^{\,2} / \dot{m}$, where U_b is the inflow velocity at building height (7 m/s) and $\dot{m}$ is the tracer release rate (kg/s).