Two-Dimensional Simulation of Air Pollution Distribution over an Urban Canyon

SAEED-REZA SABBAGH-YAZDI, ABBAS HADIAN
KN Toosi University of Technology,
Department of Civil Engineering,
No.1346 Valiasr Street, 19697- Tehran IRAN

and

NIKOS E. MASTORAKIS
Military Insitutes of University Education (ASEI)
Hellenic Naval Academy
Terma Chatzikyriakou 18539, Piraues, GREECE

Abstract:
Development of numerical solution methods, has presented considerable contribution in this respect. In this paper, the continuity equation and the equation of motion for convective dominated flow are coupled with equation of pollution concentration transport. This numerical model presents a system of simple convective equations. A Cell Vertex Finite Volume Method is applied for solving the governing equations on triangular unstructured meshes. Using unstructured meshes provides great flexibility for modeling the flow in arbitrary and complex geometries, such as urban environments. The equation of continuity is simultaneously solved with convective equations in a coupled manner using Artificial Compressibility technique. In order to verify the model, numerical model results are compared with available experimental measurements for pollutant dispersion in an urban street canyon. The efficiency of the developed computer code is demonstrated by its application to simulation pollutant transport in urban environments.

Keywords:
Numerical Simulation, Air Pollution Distribution, Finite Volume Method, NASIR[1] Flow Solver

[1] Numerical Analyzer for Scientific and Industrial Requirements

1 Introduction

The production of high performance digital computers and development of efficient numerical modeling techniques have led to the development of powerful Computational Fluid Dynamics (CFD) models for solution of problems concerning fluid flow. Hence, the computer simulation of complicated flow cases has become one of the interesting areas of the research works by development of efficient and accurate numerical methods suitable for the complex solution domain. The control over properties and behavior of fluid flow and relative parameters are the advantages offered by CFD witch make it suitable for the simulation of the applied problems.

The governing equation of pollution phenomena is formed by both transport and diffusion terms. However, definition of the coefficient of dispersion for every single type of pollution is not an easy task and the solution of diffusion terms (high order spatial derivatives) is time consuming. Therefore, for the cases where the diffusion part of this equation can be ignored, considerable simplification in both equation and solution procedures are gained.

Artificial Compressibility technique for steady state equations is used to overcome the problem of imposing zero velocity divergence to the equation of motion. The governing conservative equations, which contain only first order spatial derivatives of the conserved variables, are solved using Cell Vertex Finite Volume Method on triangular unstructured meshes. Proper terms of artificial dissipation are used to stabilize the numerical solution procedure. An edge-base algorithm covered the efficiency shortcoming of the unstructured mesh data processing [1].

In this paper, the two dimensional pollution transport module of NASIR (Numerical Analyzer for Scientific and Industrial Requirements) is introduced and applied for a reel case, after verification.

As verification, the solution of continuity equations and fluid motion coupled with pollution concentration transport equation are used to simulate two-dimensional test cases for smoke transport in urban canyons which have available experimental measurements. In order to present ability of the numerical model dealing with real cases, the transport of pollution from a single source or several sources, point sources or line sources, due to wind effect in urban environment with geometrical complexities are simulated. The efficiency and accuracy of the developed model presents considerable achievement in numerical solution of engineering problems.

2 Governing Equations

For flow problems (Mach < 0.3), since density is constant, the fluid flow is considered incompressible. In high Reynolds number flow, the boundary layer is thin and very close to solid walls and the effects of viscosity are confined to a thin layer and ignorable in scale of flow field [2]. Outside the boundary layer convection dominated condition is acceptable for the fluid flow behavior. Considering isothermal condition for the flow problem decouples the energy equation from the governing equations. Hence, for the low velocity isothermal flow problems with high Reynolds numbers, the incompressible Euler equations are considered as the set of governing equations which consist of the continuity equation and momentum equations in Cartesian directions.

In the incompressible form of Euler equations, time derivative term eliminates from continuity equation, and this matter presents some numerical difficulties in coupled solution of the continuity and the momentum equations. For steady state incompressible problems, the Artificial Compressibility technique helps to overcome this problem [3].

It is common to use a convection-diffusion equation for solving the pollutant concentration transport in fluid flow field. This equation consists of two convective and diffusive parts [4]. For high Reynolds number flows, the effect of diffusion is negligible in comparison with the convection due to velocity field. This is the usual case for the pollution transport due to wind flow in urban environments.

In this work, the governing equations are simplified by considering the wind flow in urban fields as incompressible, convection dominated, isothermal flow and by neglecting the turbulence effects and the effects of chemical reactions and diffusion for the pollutant concentration. The conservative form of the governing equations are written in vector form as follows:

$$\frac{\partial W}{\partial t}+(\frac{\partial F}{\partial x}+\frac{\partial G}{\partial y})=S \qquad (1)$$

$$W=\begin{pmatrix} p/(\rho_o\beta^2) \\ u \\ v \\ C \end{pmatrix}, \quad F=\begin{pmatrix} u \\ u^2+p/\rho_o \\ uv \\ uC \end{pmatrix},$$

$$G=\begin{pmatrix} v \\ uv \\ v^2+p/\rho_o \\ vC \end{pmatrix}, \quad S=\begin{pmatrix} 0 \\ 0 \\ 0 \\ s_c \end{pmatrix}$$

Where, w represents the conserved variables and F and G are vectors of convective fluxes of w in x and y directions, respectively. u and v the components of velocity, p pressure and C volumetric percentage of concentration are four dependent variables. Here, ρ_o represents the constant density and S is source term related to pollution production. This term has non-zero quantity at sources of pollution and keeps zero at all other points. The parameter β is introduced using the analogy to the speed of sound in equation of state of compressible flow, by application of the Artificial Compressibility technique [5].

3 Numerical Formulations

The governing equations are discretized by the application of cell vertex (overlapping) scheme of the finite volume method. This method gives the following formulation [3]:

$$\frac{(W_i^{n+1}-W_i^{n})}{\Delta t}=-\frac{1}{\Omega_i}.\sum_{k=1}^{N_{sides}}(\overline{F}\Delta y-\overline{G}\Delta x)_k^n+S \qquad (2)$$

Where Wi represents conserved variables at the center of control volume Ωi. $\overline{F}$ and $\overline{G}$ are the mean values of fluxes on the control volume boundary sides. Superscript n and n+1 show nth and the n+1th time stage. Δt is the time step (proportional to the minimum mesh spacing) applied between time stages n and n+1. In present study, a three-stage Runge-Kutta scheme is used for stabilizing the computational process by damping high frequency errors, which this in turn, relaxes CFL condition [6].

The convective equations such as Euler equations do not provide any dissipation mechanism that would eliminate the oscillations near the high gradient regions. In order to damp unwanted numerical oscillations associated with the explicit solution of Convective term in the above algebraic equation, $C(W_i)=\sum_{k=1}^{N_{sides}}(\overline{F}\Delta y-\overline{G}\Delta x)_k^n$, a fourth order artificial dissipation term, $D(W_i)=\varepsilon\sum_{j=1}^{N_{edges}}\lambda_{ij}(\nabla^2 W_j-\nabla^2 W_i)$ is added to above algebraic formula in which λ_{ij} is a scaling factor and is computed using the maximum value of the spectral radii of every N_{edgess} edges connected to node i and $1/256\le\varepsilon\le 3/256$. Here the Laplacian operator at every nodes i, $\nabla^2 W_i=\sum_{j=1}^{N_{edges}}(W_j-W_i)$, is computed using the variables W at two end nodes of all N_{edge} edges (which meet at node i). The revised formula, which preserves the accuracy of the numerical solution is written in the following form [1].

$$W_i^{n+1}=W_i^{n}-(CFL\ \Delta t)\left\{\frac{1}{\Omega_i}[C(W_i)-D(W_i)]+\ S\right\} \qquad (3)$$

In the above equation, the quantities W at each node is modified at every time step by adding a residual term of $R(W_i)=\Delta t[C(W_i)-D(W_i)]/\Omega_i+\Delta t\,S$ which is computed using the quantities W at the nodes of boundary sides of the control volume Ωi. Hence, the edges are referred to all over the computation procedure. Therefore, it would be convenient to use the edge-base data structure for definition of unstructured meshes. It has been shown that using the edge-base computational algorithm reduces the number of addressing to the memory, and therefore, provides a 50% saving in computational CPU time [1].

Two types of boundary conditions are applied in this work; flow and solid wall boundary conditions. The flow boundary condition is developed from the similarity to the one dimensional characteristic theory for the first order wave equations. From this theory, the prorogation directions are defined according to the sign of the system waves of the incompressible convection dominated equations (Eigenvalues of the Jacobian matrix of the continuity and motion equations). The values of related quantities are imposed wherever the characteristics enter the computational domain. Conversely, at boundaries where characteristics leave the domain, nodal value of related quantities are determined from the interior solution domain. For incompressible flow, at the inflow boundaries free stream, values of u and v are imposed and p is extrapolated from inside domain nodes, and at the outflow boundaries free stream, p is imposed and u and v are extrapolated from inside domain nodes. The nodal value of concentration variable C at flow boundaries follows the roles for u and v. Since the flow is considered convection dominated, at the solid wall nodes, the component of velocity vector normal to the solid boundary edges are set to zero [2].

4 Pollution Transport Verification

The pollution transport module of NASIR (Numerical Analyzer for Scientific and Industrial Requirements) finite volume flow solver is verified in this section for this goal, numerical simulation of a laboratory set up (figure 1) is performed for modeling the pollution dispersion in an urban street canyon due to wind flow [7]. This case is used to evaluate the performance of the numerical convection dominated flow solver to simulate some experimental cases of concentration transport.

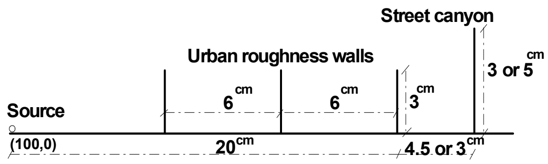


Figure 1: The experimental wind tunnel which its measurements are used for the verification (the pollutant source position and the canyon).

The test was carried out in a wind tunnel with 0.34m×0.34m square cross section and 2m length. Wind flow is created by an axial fan placed at the end of the diffuser. The incoming flow is modified by a grid placed at the entrance of the test section. The free stream velocity is $v\infty$ =1.7 m/s and is controlled by a fan anemometer. The velocity corresponds to mean flow Reynolds number of 3400, based on the height upstream canyon wall. In this Reynolds number the flow pattern is independent of viscous effects [8]. A simple configuration is applied to the canyon and urban roughness modeling. Two thin metal walls (thickness was about 1mm) parallel to each other and perpendicular to air flow which extended over the whole width of the test section, are used to model the two-dimensional street canyon and its urban roughness is simulated by putting similar street canyon upstream of the test street canyon. To model the pollution source an orifice for transfusing smoke to wind tunnel was installed on the floor (in the middle of the tunnel). The measured values consist of normalized concentration at three different levels of 5mm, 15mm and 35mm above the tunnel floor. The concentration intensity of all measurements has been normalized using a reference value measured 5mm above the top of the upstream wall.

The set up is used for three different conditions. In the first case a (reference), both of the canyon walls were 3cm in height with a 4.5cm distance and the urban roughness (upstream walls) was eliminated. In the second case b (urban roughness) the geometry of the canyon itself remained unchanged, but two 3cm high walls were put at distance of 6cm and 12cm upstream of the test street canyon (as the urban roughness). In the third case c (heights ratio) the same conditions of the reference conditions have been applied, but only height of the second canyon wall has been increased to 5cm.

The close view of the computed velocity vectors and contours of the concentration percentage are presented in figures 2 and 3, respectively. Numerical solution results of the concentration percentage are compared with experimental measurements in figures 4,5 and 6. Note that, the computed results and experimental measurements for the sections 35mm and 15mm above floor of the canyon are in good agreement with the experimental measurements (with less than 4% errors in figures 4 and 5). While in the section 5mm above the floor of the canyon (close to the tunnel floor) errors increase (figure 6), particularly for the reference case a in which the average error increases up to 20%.

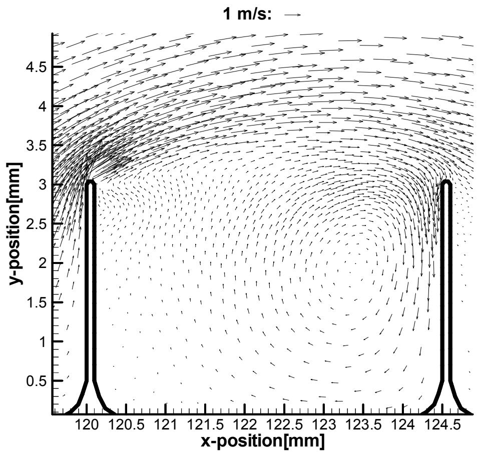


Figure 2.a: Computed velocity vectors Case a - reference

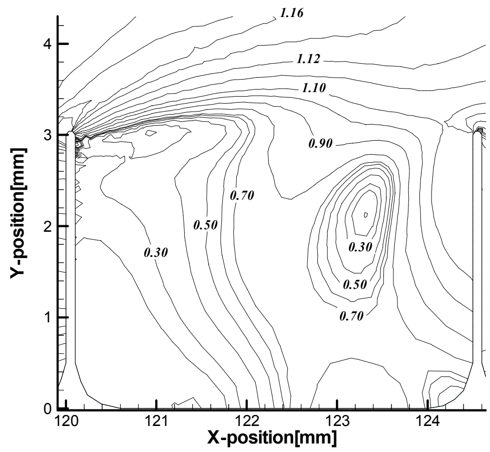


Figure 3.a: Computed velocity contours Case a – reference

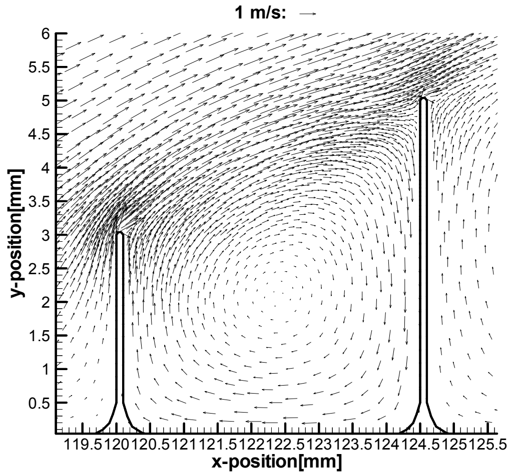


Figure 2.c: Computed velocity vectors Case c - heights ratio

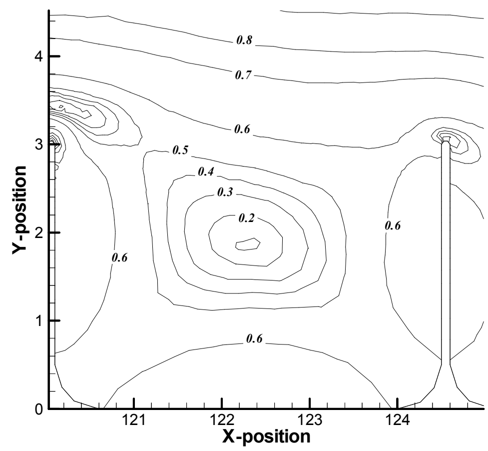


Figure 3.b: Computed velocity contours Case b - urban roughness

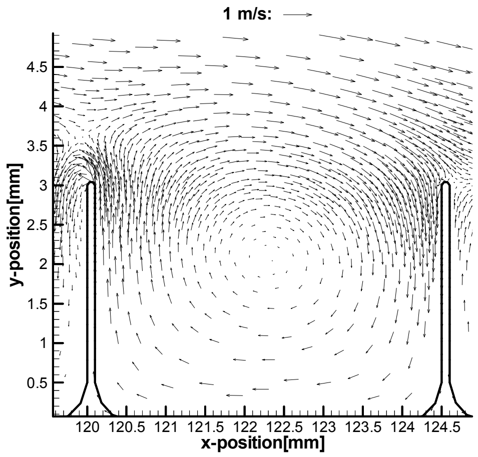


Figure 2.b: Computed velocity vectors Case b - urban roughness

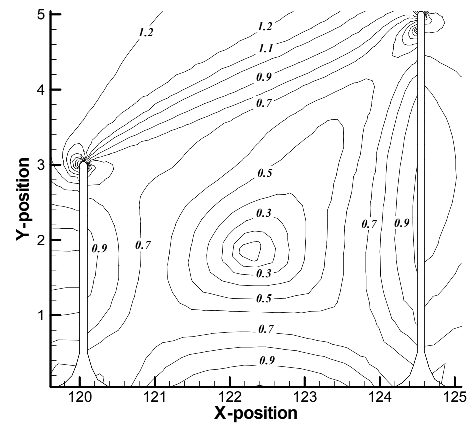
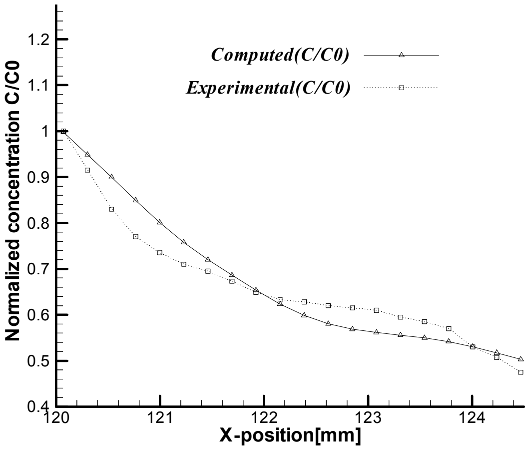
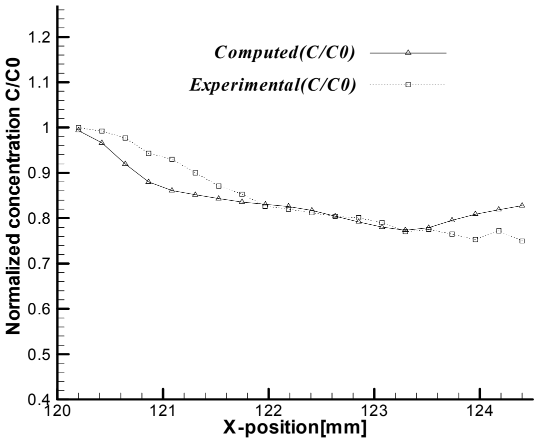


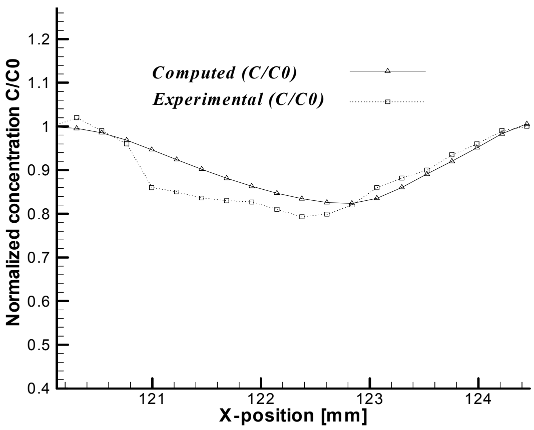
Figure 3.c: Computed velocity contours Case c - heights ratio



Case a: reference

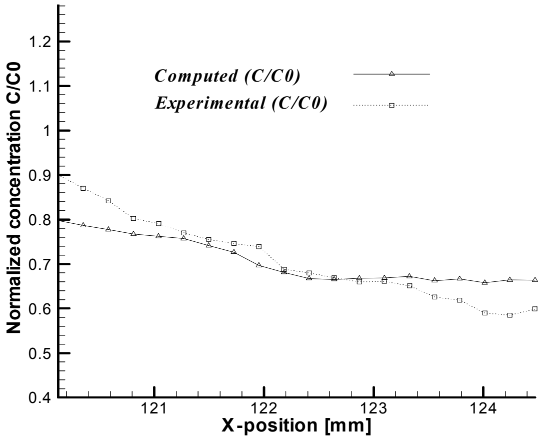


Case b: urban roughness

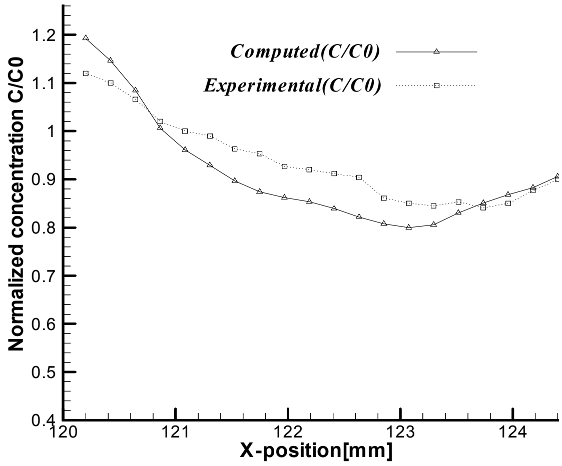


Case c: heights ratio

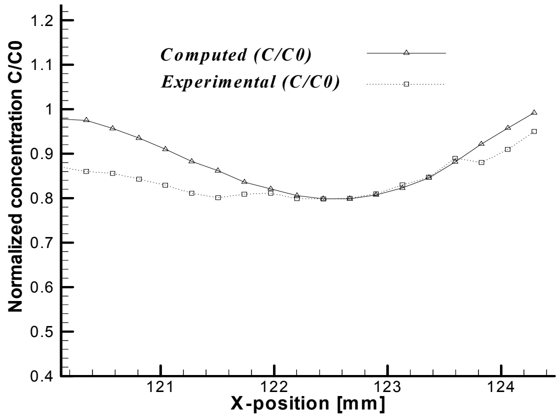
Figure 4: Horizontal profile of normalized concentration at 35mm height from the canyon floor in three cases



Case a: reference

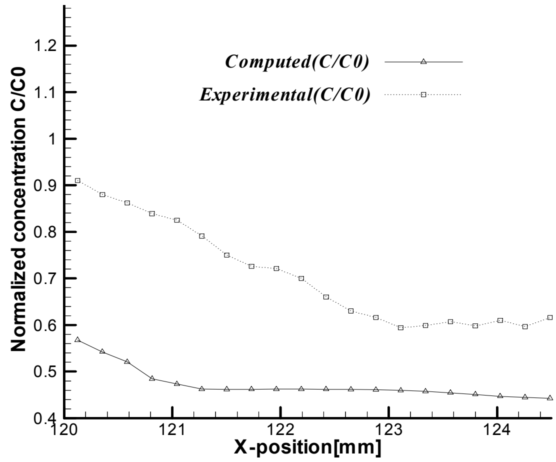


Case b: urban roughness

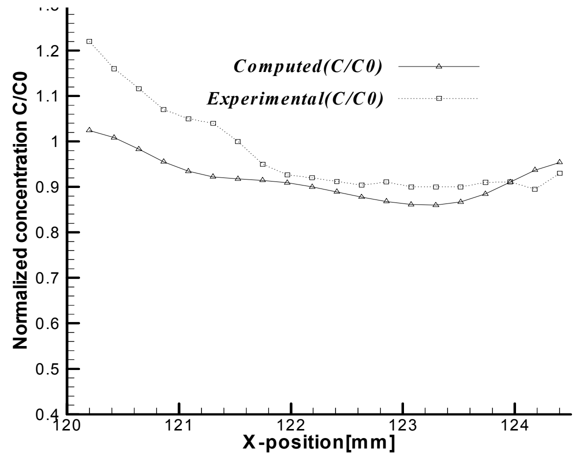


Case c: heights ratio

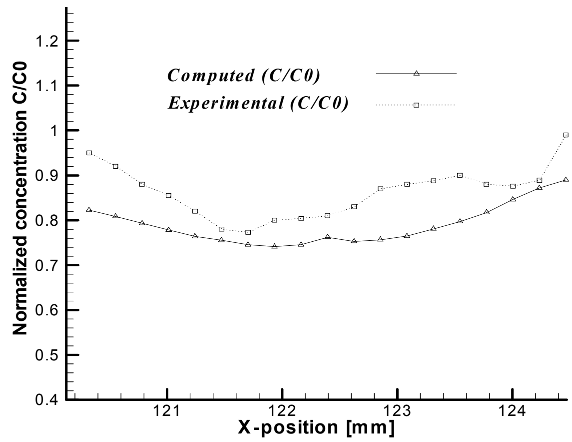
Figure 5: Horizontal profile of normalized concentration at 15mm height from the canyon floor in three cases



Case a: reference



Case b: urban roughness



Case c: heights ratio

Figure 6: Horizontal profile of normalized concentration at 5mm height from the canyon floor for three cases

These are mainly because of neglecting the diffusive terms in the equations of motion and the transport equation, which is not consistent with the condition in low velocity part of the flow field (see figure 2.a). In such a condition the thickness of boundary layer increases, and hence, the effects of fluid viscosity, turbulent of flow field and natural diffusion of the concentration are important. Therefore, development proper parts to the present algorithm for modeling the aforementioned phenomena decrease errors.

It can be concluded that, for mentioned conditions in the regions of the flow field that the boundary layer has negligible thickness, the use of transport model would provide acceptable results.

5 Conclusions

According to verification of the computer model, the model results are acceptable in a wide domain of flow fields in which the effect of viscosity and turbulent can be neglected. The two-dimensional transport numerical simulation has been successfully performed. For the considered set of urban canyon experimental test cases, the result of the developed model is in good agreements with the measured data, except close to solid walls in low velocity flow regions.

References

[1] S-Yazdi, R.S. Face-Base Solution of Incompressible Convection dominated Equations on Unstructured Meshe, International Conference of Aero Space, Sharif University of technology, Iran, 2001.

[2] Reddy, J.N. & Gatling, D.K. The Finite Element Method in Heat Transfer and Fluid Dynamics, CRC press, 2001.

[3] Dreyer.j. Finite Volume Solution to the Steady Incompressible Euler Equation on Unstructured Triangular Meshes, M.sc Thesis, MAE Dept., Princeton University, 1990.

[4] Briter R. & Griffiths, Dense Gas Dispersion, Elsevier Scientific, 1982.

[5] S-Yazdi R.S. Simulation at the Incompressible Flow Using the Artificial Compressibility Method, Ph.D Thesis, University of Wales, Swansea, 1997.

[6] Jameson A., Schmidt W. & Turkel E. Numerical Solution of the Euler Equations by Finite Volume Method using Runge Kutta Time Stepping Schemes. AIAA pp.1259-1281, June 1981.

[7] Gerdes F. & Olivari D. Analysis of Pollutant Dispersion in an Urban Street Canyon. Journal of Wind Engineering and Industrial Aerodynamics, Vol.82, pp.105-124, 1999.

[8] Meronoy R.N., Pavageau M., Rafalidis S. & Schatzmann M. Study of line source characteristics for 2-D physical modeling of pollutant dispersion in street canyons. Journal of Wind Engineering and Industrial Aerodynamics, Vol.62, pp.37-56, 1996.