NUMERICAL ANALYSIS OF CAR EXHAUST POLLUTANT CONCENTRATION IN UNDERGROUND CAR PARK

Lucyna Brzozowska and Krzysztof Brzozowski
University of Bielsko – Biała, Willowa 2, 43-309 Bielsko – Biała Poland
Tel./fax (033) 8100095 e-mail: lbrzozowska@pb.bielsko.pl

Abstract

In our paper we would like to present the computational model and programmes which enable calculations of the concentration of pollutants in underground garage caused by assumed scenarios of cars motion for given parameters of building geometry and ventilation. The finite volume method (FVM) has been applied in order to discretise the differential equations. Both problems: velocities of air distribution and pollutants dispersion are modelled by FVM. The special package of programmes has been developed for the case analysed.

1. Introduction

Modelling of pollutant dispersion in indoor environment needs the complex 3D computational fluid dynamics models to be applied. The problem becomes more complicated in the case when sources of pollutants are moving in space and time. These problems occur when underground garage is considered. Each car is a moving source of pollutants and the amount of exhausted pollutants depend on vehicle speed and ambient temperature. The data can be described by individual characteristics of car engines obtained by measurements in European scientific programmes. For the case when cold start of vehicles is considered (in that case the emission of pollutants is extremely large), the individual characteristics of car engines can by obtain by modification of characteristics obtained from measurement for hot engines. In order to solve the dispersion equations we should determine the flow of air in the car park.

2. Numerical modelling of air speed field

If we assume that air can be treated as a viscous and incompressible fluid, and if we disregard the effect of temperature on flow, the differential equations describing fields of pressures and speeds can be written as follows [3]:

\[
\sum_{l=1}^{3} \frac{\partial U^{(l)}}{\partial x^{(l)}} = \frac{\partial U^{(1)}}{\partial x^{(1)}} + \frac{\partial U^{(2)}}{\partial x^{(2)}} + \frac{\partial U^{(3)}}{\partial x^{(3)}} = 0
\]  

(1)

\[
\frac{\partial U^{(1)}}{\partial t} + \sum_{l=1}^{3} \frac{\partial}{\partial x^{(l)}} (U^{(1)} U^{(l)}) - \mu \sum_{l=1}^{3} \frac{\partial}{\partial x^{(l)}} \left( \frac{\partial U^{(1)}}{\partial x^{(l)}} \right) + \frac{1}{\rho} \frac{\partial p}{\partial x^{(1)}} = g^{(1)}
\]  

(2.1)

\[
\frac{\partial U^{(2)}}{\partial t} + \sum_{l=1}^{3} \frac{\partial}{\partial x^{(l)}} (U^{(2)} U^{(l)}) - \mu \sum_{l=1}^{3} \frac{\partial}{\partial x^{(l)}} \left( \frac{\partial U^{(2)}}{\partial x^{(l)}} \right) + \frac{1}{\rho} \frac{\partial p}{\partial x^{(2)}} = g^{(2)}
\]  

(2.2)

\[
\frac{\partial U^{(3)}}{\partial t} + \sum_{l=1}^{3} \frac{\partial}{\partial x^{(l)}} (U^{(3)} U^{(l)}) - \mu \sum_{l=1}^{3} \frac{\partial}{\partial x^{(l)}} \left( \frac{\partial U^{(3)}}{\partial x^{(l)}} \right) + \frac{1}{\rho} \frac{\partial p}{\partial x^{(3)}} = g^{(3)}
\]  

(2.3)

where:
Equation of continuity (1) and equations of momentum (Navier-Stokes) (2) represent a system of four non-linear partial differential equations. Solution of equations (1) and (2) normally proceeds as follows: stage 1 – the equations are transformed into a system of algebraic and ordinary differential equations with respect to variable \( t \). This stage, which eliminates derivatives with respect to spatial variables \( x^{(1)}, x^{(2)}, x^{(3)} \), is called the stage of discretising equations. In stage 2 – the system of algebraic and differential equations is solved by reducing the task to the problem of solving systems of algebraic equations.

In view of easy interpretation of boundary conditions, we shall use the finite volume method or rather its modification, which in Polish sources is called the control domain method (CDM) [3,6]. In this method the integral form of continuity and momentum equations is not directly used, as in the finite volume method, but their differential form (1) and (2). This specific and ingenious method of discretisation eliminates the intermediate stage found in classical FVM which transforms the equations at the centre of volumes into equations at the faces [11]. In order to ensure a high degree of accuracy and stability of numerical solutions, we use the Crank-Nicholson method. At this stage there are usually problems of iterative solving of equations after discretisation (which is connected with approximation of the field of pressures) and solving large systems of algebraic linear equations.

3. Discretisation of Navier-Stokes equations

The domain forms is a rectangular prism. Air motion is created by a number of inlets, in which the speed of air in the direction normal to the surface is defined, and outlets which ensure exchange of air. Within the domain an element is distinguished whose geometrical centre has the coordinates \( x^{(1)}, x^{(2)}, x^{(3)} \). Before defining its dimensions, we shall describe the general method of denoting points and elements.

If we investigate axis \( r \), then, assuming that the length of the side of the domain in direction \( r \) is \( L^{(r)} \), denotation as in Fig.1 is used.

![Fig. 1. Denotation when dividing interval \((0, L^{(r)})\) into subintervals \((r = 1,2,3)\)]
Interval $[0,L^{(r)}]$ is divided into $n_r$ subintervals with lengths $\Delta_i^{(r)} - \Delta_{i+1}^{(r)}$. In the centre of the subintervals points $x_i^{(r)} - x_{i+1}^{(r)}$ are located. The ends of the subinterval whose centre is $x_i^{(r)}$ are denoted $x_{i-\frac{1}{2}}^{(r)}$ and $x_{i+\frac{1}{2}}^{(r)}$. Subintervals may be of equal or different length.

We try to find

$$U_{i,j,k}^{(1)} = U^{(1)}(t, x_{i\frac{1}{2}}, x_{j\frac{1}{2}}, x_{k\frac{1}{2}}) \quad (3.1)$$

for $i = 1, \ldots, n_1 - 1, j = 1, \ldots, n_2, k = 1, \ldots, n_3$,

$$U_{i,j,k}^{(2)} = U^{(2)}(t, x_{i\frac{1}{2}}, x_{j1}, x_{k\frac{1}{2}}) \quad (3.2)$$

for $i = 1, \ldots, n_1, j = 1, \ldots, n_2 - 1, k = 1, \ldots, n_3$,

$$U_{i,j,k}^{(3)} = U^{(3)}(t, x_{i\frac{1}{2}}, x_{j\frac{1}{2}}, x_{k1}) \quad (3.3)$$

for $i = 1, \ldots, n_1, j = 1, \ldots, n_2, k = 1, \ldots, n_3 - 1$.

After integrating equations for respective control volumes, they have the discrete form:

$$BU = h \quad - \text{continuity equation}, \quad (4.1)$$

$$\dot{U} + f(U) + A p = 0 \quad - \text{Navier-Stokes equation}. \quad (4.2)$$

Matrices $A$ and $B$ are matrices with constant coefficients [2].

The difficulty of solving equations (4) is due to:

1. non-linearity of equation (4.1),
2. vector $p$ is not known while solving equation (4.2).

In the paper we use a procedure which enables equations (4.2) to be integrated according to the Crank-Nicholson scheme. This iterative procedure is a modification of the PISO algorithm [11] in order to determine vector $p$.

4. Pollutant dispersion equation

The equation of pollutant dispersion can be written in the following form [2]:

$$\frac{\partial \phi}{\partial t} + \sum_{m=1}^{3} \left[ \frac{\partial}{\partial x^{(m)}} \left( f^{(m)} \phi \right) - \frac{\partial}{\partial x^{(m)}} \left( \mu^{(m)} \frac{\partial \phi}{\partial x^{(m)}} \right) \right] + \sigma \phi = f \quad (5)$$

where:
- $\phi$ – concentration of pollutants at coordinate point $(x^{(1)}, x^{(2)}, x^{(3)})$,
- $U$ – vector of air speed with components: $U^{(1)}, U^{(2)}, U^{(3)}$,
- $\mu^{(1)}, \mu^{(2)}, \mu^{(3)}$ – eddy diffusivity in directions $x^{(1)}, x^{(2)}, x^{(3)}$ respectively,
- $f$ – rate of pollutant emission from source per unit volume,
- $\sigma$ – coefficient of absorption (settlement) of pollutants.
The finite volume method was used to discretise equations describing the process of pollutant dispersion [2]. This method also uses algorithms in order to obtain numerical stability. Additionally, for solving the set of ordinary differentially equations of first order, obtained after discretization of dispersion equations (5), the decomposition two-cycle scheme method was used [2,7]. The scheme used, gives an identical accuracy to the Crank-Nicholson scheme and error is therefore of the order $(\Delta t)^2$. The decomposition method radically reduces time of numerical calculations. It is also important to notice that matrices in equations solved are strip (pentadiagonal) matrices [2].

### 5. Emission Characteristics

The majority of methods of estimating vehicle emissions are based on models in which characteristics (emission in function of vehicle motion parameters) are elaborated on the basis of average emission factors as a function of:

- average speed of motion and type of motion,
- only average speed of motion.

In models of the first kind emission characteristics are elaborated as a function of average speed of motion for a defined type of dynamic motion in relation to frequency of changes of speed of motion [4,5] Such a method of estimating emissions is adopted by the HBEFA model [4], which uses defined emission characteristics of exhaust gas pollutants in relation to category of vehicle and type of motion. Results were then statistically processed in order to obtain emission characteristics as a function of average speed of motion, category of vehicle and type of motion.

In models of the second kind emission characteristics are elaborated as a function only of average speed of motion. An example of such an approach is the model developed in 1987 by European laboratories and research centres within the CORINAIR group [5]. The model, in the form of the computer program COPERT, is continually developed and modified. In 2000 the third version was published [8,9]. Data contained in the COPERT program was obtained in various drive cycles at various average speeds with various dynamics. Definition of average emission factor as a function of average speed of motion is based on determination of emission of a particular exhaust gas pollutant over the whole cycle characterised only by average speed [4]. In this model vehicles are classified in five categories: PC, LDV, HDV, buses and two wheelers. The individual categories were further sub-divided into classes according to type of engine (SI or CI), engine capacity (for cars), vehicle mass (for HDV), and emission norms met by the vehicle. Two-stroke engines and gas-powered engines were also included. SI personal cars were classified in three classes according to engine capacity and CI personal cars were classified in two classes.

Models based on the average emission factor contain algorithms which enable cold start and heating engine to be taken into account in calculations. Of interest in this regard is the methodology in the COPERT program [8]. In the COPERT model the coefficient of emission (relative emission) was defined for individual exhaust gas pollutants as a function of ambient temperature; speed of motion and length of trip immediately after cold start were not considered. Data was collected for personal cars and to a limited extent for light duty vehicles. The coefficient of emission is defined according to the following relationship

$$\eta_{e/h}(v,T) = \frac{E_c}{E_h} = Av + BT_a + C$$

where:

- $\eta_{e/h}$ – coefficient of emission,
- $E_h$ – emission in the hot phase,
\[ E_c \] – emission in the cold phase,

\[ v \] – speed ,

\[ T_a \] – ambient temperature [\( ^\circ C \)],

\[ A, B, C \] – model coefficients.

The main parameters determining emissions during cold start are:
- ambient temperature,
- speed of vehicle motion,
- length of trip (time of engine work).

Vehicle engine heating profile can be introduced according to the third parameter and it can be written in form [1]:

\[ T_r - T_a = f(t) \] (7)

where:

\[ T_r \] – engine temperature in time \( t \) elapsed from engine start.

In order to obtain function \( f(t) \) authors of [1] use a polynomial in the form:

\[ T_r - T_a = c_1(t + t') + c_2(t + t')^2 \] (8)

where:

\[ c_1, c_2 \] – experimental coefficient,

\[ t' \] – depends on the vehicle start temperature and is calculated by solving equation (8) for \( t = 0 \) and \( T_r = T_0 \) (engine start temperature).

Function \( f(t) \) can be also expressed in exponential form:

\[ T_r - T_a = c_1 \left(1 - \exp(c_2(t + t'))\right) \] (9)

Similarly, we can construct a curve, that give information about time evolution of excess emission in cold start conditions. Paper [1] suggests the following function to determine fraction of total excess emission:

\[ F_{AE}(t) = 1 - \exp(-bt) \] (10)

where:

\[ F_{AE}(t) \] – fraction of total excess emission,

\[ b \] – experimental coefficient,

\[ \Delta E = E_h(\eta_{c/b} - 1) \] – excess emission during heating.

We assume, that emission in time \( t \) after cold start of engine can be calculated as:

\[ E(t, v) = E_h(v) + \Delta E(t) \] (11)

where:

\[ \Delta E(t) \] – excess emission in function of time and \( \Delta E(t) = \Delta E \left(1 - F_{AE}(t)\right)\).
1. Numerical simulation

The problem was to determine possible distributions of pollutant concentrations in the domain of the underground car park for given ventilation conditions. Numerical analysis was made for carbon monoxide (CO). Carbon monoxide was emitted by 56 passenger cars (each of which represented a moving source of pollutants in accordance with the multi-point model) leaving different parking spaces at random every 15 seconds. The speed of each vehicle conformed to the first phase of the UDC cycle. Individual emission characteristic of CO from cars was assumed to be according to the COPERT III model [8]. The excess emissions from cars of one category after cold start were assumed also according to this model. We also apply equation (11) to determine excess emission in time after cold start in ambient temperature 5°C. The percentage distribution of cars according to pollutant emission was assumed on the basis of statistical data about new cars sold in Poland after 1996, like in [2].

The domain of the car park and initial arrangement of cars is shown in Fig. 3.

It can be observed, that in the car park cars are situated symmetrically around two roads and the car park has four gates situated at the end of each road. Cars leaves the domain in as shown by arrows in the figure. Ventilation pump system works in the ceiling of car park with total intensity of 48 cubic meter per second. The dominant component of air speed vector is \( U^{(3)} \). An example of \( U^{(3)} \) is presented in Fig.4. Calculated concentration of CO in different time points in car park on high 1.8 m is presented in Fig.5. All CO concentrations were displayed on an absolute scale \( \log \phi / \log \phi_{\text{max}} \) in relation to the maximum concentration \( \phi_{\text{max}} \) of carbon monoxide (equal 0.022 mg/m\(^3\)) calculated in the car park.
In order to evaluate health risk made by pollutant emission, it is more practicable to use information about averaged concentration of pollutant in selected time period in car park domain rather than instantaneous concentration. This kind of information enable us to optimise the arrangement of ventilators designed to generate air speed field which might reduce concentration of pollutants. Averaged calculated concentration of CO is presented in Fig. 6.
1. Conclusions

The algorithms and computer program elaborated enable us to determine concentration of pollutants in a closed (underground) car parks when the scenario of vehicle motion is assumed and emission characteristics of vehicles are known. Algorithms used to determine air speed are verified by comparison of results obtained by using professional STAR CD package. Satisfactory compatibility of results was achieved [2].

References


ANALIZA NUMERYCZNA KONCENTRACJI ZWIĄZKÓW SZKODLIWYCH SPALIN W OBRĘBIE PARKINGÓW PODZIEMNYCH