Bibliографические ссылки


Надійшла до редколегії 26.06.2017

УДК 502.3:504.5:629.33

1. Dnipropetrovsk National University of Railway Transport named after Academician V. Lazaryan

2. National Transport University of Ukraine

MATHEMATICAL SIMULATION OF POLLUTION DISPERSION IN URBAN STREET

Development of numerical model, which allows quick computation air pollution in streets from vehicles. The goal of the work is development fast calculating CFD model which takes into account the meteorological parameters, form of protection barriers near the road, emission rate of toxic gases. The developed model is based on the equation of inviscid flow and equation of pollutant transfer. Equation of potential flow is used to compute flow pattern near road in the case of protection barriers application. To solve equation for potential flow implicit difference scheme of «conditional approximation» is used. The implicit change – triangle difference scheme is used to solve equation of convective – diffusive dispersion. Numerical integration is carried out using the rectangular difference grid. Method of porosity technique («markers methods») is used to create the form of comprehensive computational region. Emission of toxic gases from vehicle is modeled using Delta function for point source. Developed 2D numerical model takes into account the main physical factors affecting the process of dispersion of pollutants near the road. The model takes into account the influence of vehicle
and protection barriers on pollutant dispersion near the road. On the basis of the developed numerical models a computational experiment was performed to estimate the influence of protection barrier form on local air pollution near the road. Developed numerical model allows to calculate the 2D flow pattern near the road where the protection barrier is used. Model allows to perform fast calculations of the air pollution near the road.

Key words: air pollution; urban street, pollution dispersion, numerical simulation.

Conclusion. Prediction of air pollution in streets caused by vehicles is a problem of great interest. It’s known that physical modeling, in this case, is very expensive [8]. For quick prediction empirical models are used [1]. These models are convenient in practice, especially when we must run many «pilot» calculations. But these models do not take into account some important properties of pollutant dispersion process in streets. The main problem is that the process of air pollution in streets takes part in the region
having comprehensive geometrical form (presence of buildings, different obstacles, etc.). The alternative way is the numerical simulation of this process. Many authors apply CFD simulation to solve the problem [1; 2; 7–10]. As a rule, to obtain flow pattern in streets foreign authors use Navier – Stokes equations (this is the model of viscous fluid) coupled with turbulent models. Very often commercial codes are used for this purpose. Worthy of note, that application of Navier – Stokes equations needs application of very fine computational grid during the computational experiment to simulate in detail the process of vortexes formation and their dispersion and interaction in the region. Using the model of viscous fluid we must use very fine grid inside the boundary layers. This is a real problem if we have big dimensions of the buildings, obstacles in streets. So, in case of Navier – Stokes equations application it is necessary to use powerful PC and every computational experiment consumes much time. This is not convenient when we must run a lot of practical calculations considering different scenario of air pollution in streets and, especially, when we try to find the effective protection measures because in this case we must consider many alternative variants of protection. In this case it would be better to split the study in two steps. At the first step we may find the “satisfying” variant using numerical model which does not consume much time and not take into account some physical features of the process. After that, at the second step, we may use more powerful model to compute in detail the variant of protection which has been chosen. So, for quick calculations at the first step it is important to have CFD models which consume not much computational time but they allow to take into account such important features as obstacles, emission rate, etc.

**Goal of the work.** The goal of the this paper is development a numerical model for quick computing of the local air quality near roads.

**Statement of the main material. Aerodynamic equation.** To simulate the wind pattern near the road we use model of potential flow. In this case the governing equation is [5]:

\[
\frac{\partial^2 P}{\partial x^2} + \frac{\partial^2 P}{\partial y^2} = 0,
\]

(1)

where \( P \) is the potential of speed.

The wind velocity components are calculated as follows:

\[
u = \frac{\partial P}{\partial x}, \quad v = \frac{\partial P}{\partial y}.
\]

(2)

Boundary conditions equation (1) are discussed in [ibid.]. To perform numerical integration of this equation rectangular grid was used.

To solve equation of potential flow (1) we used the difference scheme of «conditional approximation».

**Pollutant Transport Equation.** To simulate the pollutant dispersion near road equation of convective – diffusive transfer is used [1; 3; 5; 6]:

\[
\frac{\partial C}{\partial t} + \frac{\partial u C}{\partial x} + \frac{\partial v C}{\partial y} = \text{div} (\mu \text{grad} C) + \sum_{i=1}^{N} Q_i(t) \delta(x - x_i) \delta(y - y_i),
\]

(3)
where $C$ is mean concentration $C(x,y) = \frac{1}{W} \int_0^W C(x,y,z)dz$, $W$ is width of the computational region; $u$, $v$ are the wind velocity components; $\mu = (\mu_x, \mu_y)$ are the diffusion coefficients; $Q_i$ is rate of emission; $\delta(x-x_i), \delta(y-y_i)$ – are Dirac delta function; $t$ is time.

Initial and boundary conditions for Eq.3 are described in [1; 3; 5].

Before solving Eq.(3) we made it’s physical splitting into the sequence of three equations. These are the following equations:

$$\frac{\partial C}{\partial t} + \frac{\partial u C}{\partial x} + \frac{\partial v C}{\partial y} = 0,$$

$$\frac{\partial C}{\partial t} = \frac{\partial}{\partial x}\left(\mu_x \frac{\partial C}{\partial x}\right) + \frac{\partial}{\partial y}\left(\mu_y \frac{\partial C}{\partial y}\right),$$

$$\frac{\partial C}{\partial t} = \sum Q_i(t)\delta(r-r_i),$$

where $\delta(r-r_i)$ is Dirac delta function; $r_i = (x_i, y_i)$ are the coordinates of the point source.

The first equation in (4) describes pollutant transfer along trajectories. The second equation in (4) describes the diffusive dispersion of pollutant. The third equation in (4) describes concentration change under the action of source $Q$.

To solve the first and the second equations in (4) the implicit change – triangle difference scheme was used [1; 5]. To solve the third equation from (4) Euler method was used [4].

Numerical integration of difference equations is performed using rectangular grid. Values of $P$, $C$ are determined in the centers of computational cells, values of $u$, $v$ are determined at the sides of the computational cells. For coding difference equations we used FORTRAN language.

**Results.** Numerical model was used to compute $CO$ concentrations near road with curb (scenario 1, fig.1) and near the road with curb and barrier (scenario 2). This barrier had a «long wing» the length of which is 0,8m. «Body» of the vehicle is represented as rectangular. Its form and form of the curb, barrier is represented in numerical model using «markers» (porosity technique). Outlet opening of the vehicle is a passive source of emission. It means that we don’t take into account speed of gases which move from it. Arrow indicates the wind direction.
Fig. 1. Sketch of computational region (the first scenario):
1 – vehicle; 2 – source of emission (outlet opening); 3 – plume; 4 – curb

Fig. 2. Sketch of computational region (the second scenario):
1 – vehicle; 2 – source of emission (outlet opening); 3 – plume; 4 – curb; 5 – barrier; 6 – «short wing»

Results of numerical simulations are shown in fig. 3 – 5. Fig. 3 represents CO concentration field near the curb without barrier. In fig. 4 computed CO concentration near the curb with barrier is shown.

In table we present computed CO concentration above the curb (height $h=1.7m$). For data in table, position $x=0$ corresponds to barrier location (fig. 2, point A).
Fig. 3. Computed CO concentration (no barrier at the curb)

Fig. 4. Computed CO concentration (barrier with «long wing» at the curb)

### Computed CO concentration at height = 1.7m

<table>
<thead>
<tr>
<th>Distance from point A</th>
<th>Concentration (scenario: no barrier)</th>
<th>Concentration (scenario: long wing)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.6 m</td>
<td>0.92 mg/m³</td>
<td>0.87 mg/m³</td>
</tr>
<tr>
<td>1.0 m</td>
<td>0.92 mg/m³</td>
<td>0.81 mg/m³</td>
</tr>
<tr>
<td>1.4 m</td>
<td>0.90 mg/m³</td>
<td>0.77 mg/m³</td>
</tr>
<tr>
<td>1.8 m</td>
<td>0.89 mg/m³</td>
<td>0.73 mg/m³</td>
</tr>
</tbody>
</table>
As we can see from table application of barrier having «long wing» allows to reduce CO concentrations near the road.

Worthy of note that computational time was about 5 sec for each scenario.

Conclusions. Numerical model for estimating the level of atmospheric air pollution near roads is proposed. Proposed numerical model allows to predict level of pollution with account of geometrical form of vehicle, curb, barriers near the road, intensity of emission rate. The solution of the aerodynamic problem is based on the numerical integration of equation for potential flow. This allows to perform quick calculation of wind pattern near road using PC which are available now in Ukraine. To predict toxic gases concentrations near road convective–diffusive equation is used. Numerical integration of this equation is performed using implicit difference scheme. Using the developed numerical model some numerical experiments were performed to study the influence of barrier form on intensity of local contamination near road.

Further improvement of the model should be carried out in the direction of creating a 3D numerical model.

Bibliographic references


Надійшла до редакції 26.06.2017