Simulation of turbulent flow over the hill

A Y Kurbanalieva, A B Turganbaevab, K T Berdibekovac and K A Bokoevd

1 Osh State University, Osh, Kyrgyzstan
2 Osh State Pedagogical University, Osh, Kyrgyzstan
3 Osh Technological University, Osh, Kyrgyzstan
4 E-mail: kurbanalieva@rambler.ru

Abstract. An airflow around a hilly area is simulated by numerically solving the Reynolds-averaged Navier-Stokes equations within the open OpenFOAM 2006 package. To simulate turbulence, standard turbulence models implemented in OpenFOAM 2006 were used together with wall functions. The results of numerical calculations were compared to the numerical data obtained using the CFL3D and FUN3D packages.

1. Introduction

In such a mountainous country as Kyrgyzstan, where large areas of land are covered with continuous mountains and valleys, the separation of flows behind the mountains and recycling in the valleys are an important property of the flow. Since residential neighborhoods and industrial areas are mostly located on the flat bottom of the valleys, spreading of pollutant emissions to the outside atmosphere is rather difficult under certain meteorological conditions. Pollutants enter recirculation zones in the valleys. It is clearly impossible to reliably forecast pollutant dispersion without detailed and accurate information about the flow field. This work discusses the prediction of the flow field in mountainous areas.

The prediction of the field of air wind in complex areas with hills and valleys provides important information for assessing problems related to the air wind, such as wind loads on structures, the location of airports, power plants, industrial facilities, windmills and wind generators, as well as the prediction of pollutant dispersion in the atmosphere. Global circulation models or mesoscale models of the atmospheric flow are not suitable for such purposes for two reasons. First, they are based on hydrostatic approximation that provides for a balance between the pressure field and the gravity force in the vertical direction. The hydrostatic approximation may be appropriate and convenient for scales of the order of hundreds of kilometers, but this is not the case for the scales required to consider local wind actions. Pressure changes caused by inertial effects in the vertical direction (the s coordinate) cannot be neglected in local scales. Second, mesoscale models cannot resolve changes in topography in the vertical direction which are important for predicting local wind models that typically include the separation of flows and recirculation vortices [1] on surfaces with different roughness. For the purpose of predicting local wind models, micro-scale models should be used. They are usually based on the numerical solution of the Reynolds-averaged Navier-Stokes equations and the turbulence model in the boundary coordinate system corresponding to the local relief.

Therefore, the present work simulates the flow field around a two-dimensional hill using the OpenFOAM package [2] with wall functions and various two-parameter turbulence models in order to assess their relative efficiency in predicting the recirculating atmospheric wind flow. This assessment is
based on comparison with available estimated data completed using other computational hydrodynamics packages, such as CFL3D [3] and FUN3D [4].

2. Problem setting
This article considers an isothermal, incompressible and turbulent atmospheric flow over a single hill (see figure 1) that enters the computational domain from the left and leaves it to the right.

The Mach number was equal to $M=0.2$, and the Reynolds number calculated through the characteristic velocity of 69.44 m/s and the length of a computational grid in 1 m is equal to $3\times10^2$. The hill itself is located within the range from $x=0.3$ m to $x=1.2$ m. For the convenience of simulations, the horizontal parts of the hill were extended from $x=0$ to $x=1.5$ m. The maximum height of the hill is 0.05 m. The shape of the hill is defined as follows:

\[
y = \begin{cases} 
0.05 \times (\sin \left(\frac{\pi x}{0.9}\right) - \left(\frac{\pi}{3}\right))^4 & \text{for } 0.3 \leq x \leq 1.2 \\
0 & \text{for } 0.0 \leq x < 0.3 \text{ and } 1.2 < x \leq 1.5
\end{cases}
\]

The computational domain is extended up to the point $x=-25$ m in upstream direction, and up to the point $x=25$ m to the right. The height of the computational region is 5 m.

3. Mathematical model
The problem under consideration is described by the Reynolds-averaged Navier-Stokes equations (RANS) that include the continuity equation [5, 6]:

\[
\frac{\partial}{\partial x_i}(\rho U_i) = 0 
\]

and the momentum transfer equation [5, 6]:

\[
\frac{\partial (\rho U_i U_j)}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \nu \frac{\partial U_i}{\partial x_j} - \bar{u}_i \bar{u}_j \right)
\]

where $U_i$ is the component of average velocity in the direction of the coordinate $x_i$, $p$ is pressure, $\rho$ is density, $\nu$ is the viscosity coefficient.
To close equations (1) and (2), the unknown Reynolds voltages \( -\overline{u_i u_j} \) through the stress velocity \( S_{ij} \) within the Boussinesq approximation are calculated as follows [5, 6]:

\[
-\overline{u_i u_j} = 2 v_t S_{ij} - \frac{2}{3} k \delta_{ij}; \quad S_{ij} = \frac{1}{2} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right),
\]  

(3)

In the case where either \( k - \varepsilon \) or the RNG \( k - \varepsilon \) model is used, the turbulent vortex viscosity included in equation (3) is calculated according to the formula \( v_t = C_\mu k^2/\varepsilon \) through the kinetic energy of turbulence \( k \equiv \overline{u_i u_j}/2 \) and its dissipation rates \( \varepsilon \) determined through the following transfer equations [5, 6]:

\[
\frac{\partial (u_i k)}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \frac{v_t}{\sigma_k} \frac{\partial k}{\partial x_i} \right) - \overline{u_i u_j} S_{ij} - \varepsilon,
\]

(4)

\[
\frac{\partial (u_i \varepsilon)}{\partial x_i} = \frac{\partial}{\partial x_i} \left( \frac{v_t}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_i} \right) - \frac{\varepsilon}{k} \left( C_1 \overline{u_i u_j} S_{ij} + C_2 \varepsilon \right).
\]

(5)

The coefficients included in equations (4) and (5) have the values given in table 1.

| Model      | \( C_\mu \) | \( C_1 \) | \( C_2 \) | \( \sigma_k \) | \( \sigma_\varepsilon \) |
|------------|--------------|------------|------------|-----------------|--------------------------|
| \( k - \varepsilon \) | 0.09         | 1.44       | 1.92       | 1.0             | 1.3                      |
| RNG \( k - \varepsilon \) | 0.085        | 1.42-C_{1R} | 1.68       | 0.7197          | 0.7179                   |

An additional factor \( C_{1R} = \eta (1 -\eta/4.38)/(1 + 0.015 \eta^3) \) in the source term of the equation for the dissipation rate of the kinetic energy of turbulence \( \varepsilon \) in the RNG \( k - \varepsilon \) turbulence model takes into account a contribution from shear effects that are significant in recirculating flows. Here \( \eta = \sqrt{2 S_{ij} S_{ij}/k} \) \( k/\varepsilon \) is the ratio of the turbulent time scale to the average stress velocity scale. For information on other turbulence models used in this work, see [2, 5, 6].

4. Numerical model

The discretization of the computational domain is obtained by the control volume method that ensures strict observance of conservation laws, and the basic concepts of the method directly correspond to physical quantities, such as mass flow rate, flow, etc. [5, 6, 7]. The computational region is divided into a number of non-intersecting hexahedral control volumes so that each grid point is present in one control volume. The differential equation is integrated for each control volume. To calculate integrals, piecewise profiles are used as they describe a change of unknown quantity between grid points.

The quantities of \( u=69.44 \text{ m/s} \) \( p = 0 \text{ Pa} \) are set as initial conditions in internal grid points of the computational grid for velocity and pressure. As is known, no initial conditions are required for a steady-state flow. However, the OpenFOAM package requires the determination of initial velocity and pressure fields [2].

At the outlet of the channel, the fluctuation velocity scale \( \phi_f \) and the elliptical relaxation factor \( f \) are set as zero normal gradients, and pressure is set to zero pressure. For the remaining variables, the inletOutlet condition is set, which means that if the flow is directed outward at the outlet from the computational domain, then this condition is equivalent to that of the absence of the normal gradient of the quantity. In the case where this flow is directed inwards in the computational domain, this condition is similar to that with the fixedValue fixed value.

Information on the boundary conditions is given in table 2.

The boundary conditions on the solid walls of the channel for turbulent quantities – kinetic energy of turbulence \( k \) and the rate of its dissipation \( \varepsilon \) were set by means of wall functions that allow to transfer the boundary conditions directly from the walls to the first grid node [5, 6] from the wall.
To solve the equation for pressure, the \textit{GAMG} solver with \textit{DICGaussSeidel} smoothing was used. The equation for the elliptical relaxation factor $f$ was solved using the \textit{PBiCGStab} solver with the \textit{DIC} preconditioner. For the equations of the remaining variables, the \textit{PBiCGStab} solver with the \textit{DILU} preconditioner was used.

### Table 2. Used boundary conditions.

| Variable                          | Boundary  |
|----------------------------------|-----------|
| Inlet                            | Outlet    | Hill       | Symmetry          |
| Velocity, U                      | 69.44     | inletOutlet| noSlip            | symmetryPlane     |
| Pressure, p                      | zeroGradient | fixedValue 0 | zeroGradient      | zeroGradient      |
| Kinetic energy of turbulence, $k$| 1.08e-3  | inletOutlet| kqRWallFunction  | symmetryPlane     |
| Dissipation rate, $\varepsilon$  | fixedValue 469.872 | inletOutlet| epsilonWallFunction | symmetryPlane     |
| Frequency, $\omega$             | fixedValue 5220.8 | inletOutlet| omegaWallFunction | symmetryPlane     |
| Turbulent viscosity, $\tilde{\nu}$ | fixedValue 6.93e-5 | inletOutlet| fixedValue 0      | symmetryPlane     |
| Fluctuation velocity scale, $\phi_t$ | zeroGradient | zeroGradient | fixedValue, $1 \times 10^{-10}$ | symmetryPlane     |
| Elliptical relaxation factor, $f$ | zeroGradient | zeroGradient | fixedValue 0      | symmetryPlane     |

The numerical solution of nonlinear equation systems was carried out using the \textit{simpleFoam} application of the OpenFOAM package that is intended for steady-state turbulent flows and uses the well-known SIMPLE algorithm for pressure-velocity coupling [5, 6, 7]. The relative error of the convergence of iterations for all the variables under consideration was equal to $\varepsilon = 3 \times 10^{-5}$.

To increase the stability of the iterative method of solving interrelated and nonlinear algebraic equations, the following under-relaxation factors 0.9 for U and 0.7 for the remaining quantities were used.

### 5. Results of numerical calculations and discussion

The numerical results were obtained for various types of standard $k - \varepsilon$ and turbulence $k - \omega$ models, namely \textit{RNGkEpsilon}, \textit{kEpsilonLopesDaCosta}, \textit{kEpsilonPhitF}, \textit{kOmega}, \textit{kOmegaSST}. In addition, the \textit{SpalartAllmaras} model was also considered. The numerical results with corresponding data of the CFL3D and FUN3D programs are given in figures 3-6. The markers represent the numerical calculation results obtained using the CFL3D (figures 3-4) and FUN3D (figures 5-6) packages.

As can be seen from the above figures, the numerical values obtained by means of the OpenFOAM, CFL3D and FUN3D packages under the pressure coefficient $C_p$ are practically the same, except for small differences in the recirculating zone after the hill. However, according to the lift coefficient $C_l$, there are significant differences in the \textit{RNGkEpsilon} and \textit{kEpsilonLopesDaCosta} turbulence models as these models give overestimated values of the lift coefficient.
Figure 3. Pressure coefficient for the CFL3D program.

Figure 4. Lift coefficient for the CFL3D program.

Figure 5. Pressure coefficient for the FUN3D program.

Figure 6. Lift coefficient for the FUN3D program.

6. Conclusion

An airflow around a hilly area is simulated by numerically solving the Reynolds-averaged Navier-Stokes equations within the open OpenFOAM 2006 package. To simulate turbulence, standard turbulence models implemented in OpenFOAM 2006 were used together with wall functions. The results of numerical calculations were compared against numerical data obtained using the CFL3D and FUN3D packages. The numerical values of the pressure coefficient $C_p$ obtained by means of the OpenFOAM, CFL3D and FUN3D packages are practically the same, except for small differences in the recirculating zone after the hill. However, according to the lift coefficient $C_f$, there are significant differences in the
RNGkEpsilon and kEpsilonLopesdaCosta turbulence models as these models give overestimated values of the lift coefficient.

Reference

[1] Atkinson B W 1995 Introduction to the Fluid Mechanics of Meso-Scale Flow Fields, in Diffusion and Transport of Pollutants in Atmospheric Mesoscale Flow Fields (Dordrecht, Netherlands: Kluwer Academic Publishers) 1220

[2] OpenCFD Release OpenFAOM® v2006 (2006) Retrieved from: https://www.openfoam.com/news/main-news/openfoam-v20-06

[3] CFL3D (Version 6) Retrieved from: https://cfl3d.larc.nasa.gov

[4] FUN3D (Manual) Retrieved from: https://fun3d.larc.nasa.gov

[5] Ferziger J H and Peric M 2002 Computational Methods for Fluid Dynamics (Third Edition) (Berlin: Springer Verlag) 423

[6] Versteeg H K and Malalasekera W 2007 An Introduction to Computational Fluid Dynamics: the Finite Volume Method 2 (England: Pearson education Limited) 517

[7] Patankar S V 1980 Numerical Heat Transfer and Fluid Flow (New York: Hemisphere Publishing Corporation) 197