NUMERICAL SIMULATION OF 2D TURBULENT FLOW IN A BUILT-UP AREA

T. Kopáček, L. Beneš

CTU Prague, Faculty of Mechanical Engineering, Department of Technical Mathematics, Prague

Abstract

This paper deals with the numerical solution of the turbulent flow over a built-up area. Currently, our model consists of one simple house in two dimensions. The mathematical model is described by the RANS (Reynolds Averaged Navier Stokes) system of equations. The numerical approximation is based on the FVM (Finite Volume Method) with the AUSM (Advection Upstream Splitting Method) scheme and the PLR (Piecewise Linear Reconstruction) limited by the Barth-Jespersen limiter. The turbulence was modelled by k–ω TNT two–equation turbulence model. Our numerical result was compared with the experiment carried out by ARTI in their BLWT (Boundary Layer Wind Tunel).

1 Mathematical model

To model the incompressible viscous turbulent flow over a complex geometry in 2D, the system of the RANS equations completed by a proper turbulence model was used. In this case, the k-ω TNT model is considered. Using the method of the artificial compressibility, we start with the two dimensional system

$$\Gamma W_t + F_x + G_y = \nu(R_x + S_y)$$

where $\nu = \nu_M + \nu_T$, ν_M is a molecular viscosity, ν_T is the turbulent viscosity, $W = (p, u, v)^T$ is the vector of conservative variables, $\Gamma = diag(1/\beta^2, 1, 1)$ is the diagonal matrix, F and G are vectors of convective fluxes and R, S are viscous fluxes, i.e.

$$\begin{aligned} F &= (u, u^2 + p, uv)^T, \quad & R &= (0, u_x, v_x)^T, \\ G &= (v, uv, v^2 + p)^T, \quad & S &= (0, u_y, v_y)^T. \end{aligned}$$

2 Turbulent model

To model the turbulence we use the two equation k-ω TNT model. This model is based on two transport equations for k (turbulence kinetic energy) and for ω (specific dissipation rate)which are coupled with the RANS system of equations.

$$\begin{aligned} \frac{\partial k}{\partial t} + u_j \frac{\partial k}{\partial x_j} &= \tau_{ij}^R \frac{\partial u_i}{\partial x_j} - \beta^* k\omega + \frac{\partial}{\partial x_j}\left[(\nu + \sigma^* \nu_T)\frac{\partial k}{\partial x_j}\right] \\ \frac{\partial \omega}{\partial t} + u_j \frac{\partial \omega}{\partial x_j} &= \alpha \frac{\omega}{k} \tau_{ij}^R \frac{\partial u_i}{\partial x_j} - \beta\omega^2 + \frac{\partial}{\partial x_j}\left[(\nu + \sigma \nu_T)\frac{\partial \omega}{\partial x_j}\right] + \frac{1}{2}\omega \max(k_x\omega_x + k_y\omega_y, 0) \end{aligned}$$

where

$$\alpha = \frac{5}{9}, \quad \beta^* = 0.09, \quad \beta = \frac{5}{6}\beta^*, \quad \sigma^* = \frac{2}{3}, \quad \sigma = \frac{1}{2}, \quad \nu_T = \frac{k}{\omega}$$

To complete the system the Boussinesq hypothesis is used

$$\tau_{ji}^R = -2\nu_T S_{ij},$$

where τ_{ji}^R are the so called Reynolds stresses and S_{ij} are the rate of deformation tensor. For more details see e.g. [2].

3 Numerical method

The mumerical method is based on the finite volume cell centered scheme. The convective fluxes were discretized by the AUSM method and the viscous fluxes by the central way on the dual mesh. The incurred system of ODR equations was solved by the second order TVD Runge-Kutta method.

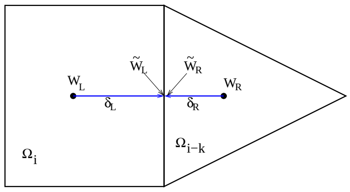
$$\iint\limits_{\Omega_i} \int_{t^n}^{t^{n+1}} \left[\Gamma \mathbf{W}_t + \mathbf{F}_x + \mathbf{G}_y - \nu \left(\mathbf{R}_x + \mathbf{S}_y \right) \right] dt d\Omega_i = 0$$

$$\frac{dW_i}{dt} = -\frac{\Delta t}{|\Omega_i|} \sum_{j=1}^{N_i} (\mathcal{F}_{ij}^{AUSM} + \mathcal{F}_{ij}^{vi}) \mathbf{n}_{ij} \Delta l_{ij}$$

$$\mathcal{F}_{ij}^{AUSM} = \left[u_n \begin{pmatrix} 1 \\ u \\ v \end{pmatrix}_{L/R} + p \begin{pmatrix} 0 \\ n_x \\ n_y \end{pmatrix} \right] \Delta l_{ij}$$

where Δt is the time step, $|\Omega_i|$ is the area of control volume Ω_i, Δl_{ij} is the length of the face common to the i^{th} and j^{th} control volumes, $\mathbf{n}_{ij}$ denotes the outer unit normal vector to the face, $\mathcal{F}_{ij}^{AUSM}$ stands for the AUSM flux and $\mathcal{F}_{ij}^{vi}$ for the viscous flux through the face, see [1]. In order to increase the accuracy of the scheme, the PLR is used in the following form

$$\begin{aligned} \tilde{\mathbf{w}}_L &= \mathbf{w}_L + \psi \left(\delta_L^T \cdot \nabla w_L + \frac{1}{2} \delta_L^T \cdot \mathbb{H} \cdot \delta_L \right) \\ \tilde{\mathbf{w}}_R &= \mathbf{w}_R - \psi \left(\delta_R^T \cdot \nabla w_R + \frac{1}{2} \delta_R^T \cdot \mathbb{H} \cdot \delta_R \right) \end{aligned}$$



where $\mathbb{H}$ is the Hesian. The symbol ψ denotes a suitable limiter. In our computation the Barth-Jespersen limiter was used, see [3].

4 Validation of the code

A test case involving turbulent flow around a flat board [3] was used to validate the code. The kinetic viscosity used in this case is $\nu_M = 3.5 \cdot 10^{-7}$. Results are shown in Figures (1)–(4).

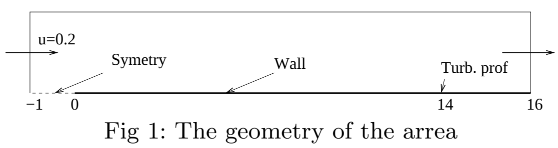


Fig 1: The geometry of the arrea

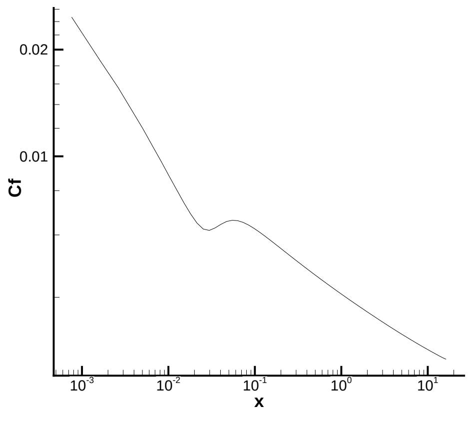


Fig 2: The distribution of a friction coeficient in log scale

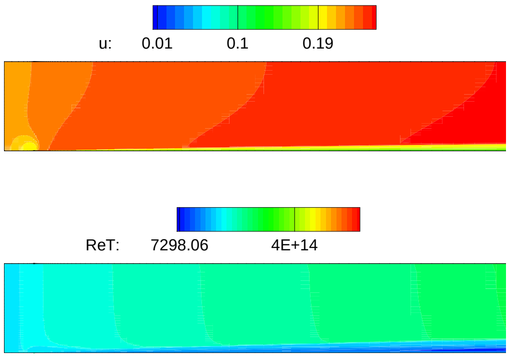


Fig 3: The stream-wise velocity component u (up) and the distribution of the turbulent Reynolds number (below)

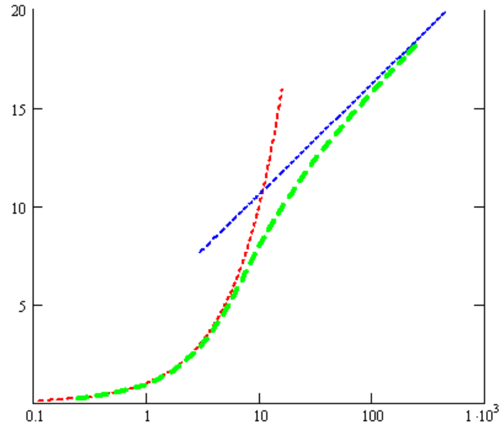


Fig 4: The conparison of the computed turbulent profile (green) with the theoretical provfile (red, blue)

5 Modeling of turbulent flow in a built-up area

The goal of this work was to obtain numerical results that could be compared with the experimental data measured by the ARTI [4]. In our test case the turbulent flow around a house is numericaly simulated. The experiment was carried out in a tunnel for simulation of the turbulent boundary layer (BLWT). Several measurements for flows over different geometries were undertaken, namely flow over the double pitched roof and the flat roof.

Numerical simulation was carried only for the flat roof. The structured quadrilateral multiblocked grid was used. The mesh Fig.(9) consists of three blocks and it's exponencilly refined close to the building and ground. On the inlet the power law was prescribed with $\alpha = 0.16$ and $u = 10$ at the level of the roof of the house. Figures (5)–(6) depicts velocity field of mean velocity obtained by the PIV method and pressure distribution on the front face of measured object. The results obtained by the numerical simulation are shown in the Fig. (7) and Fig. (8). From this, we can see good agreement between numerical and experimental data.

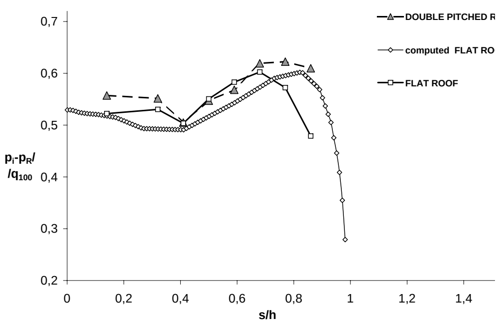


Fig 5: The comparison of the computed and measured presure distribution on the front face of building.

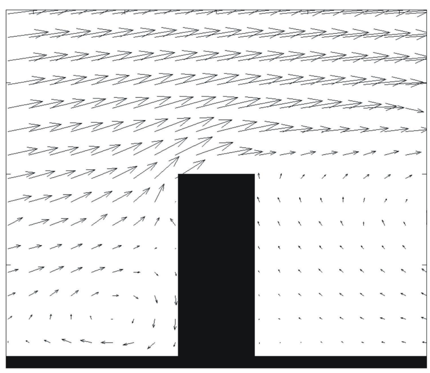


Fig 6: PIV measurement - the velocity flow field close to the building

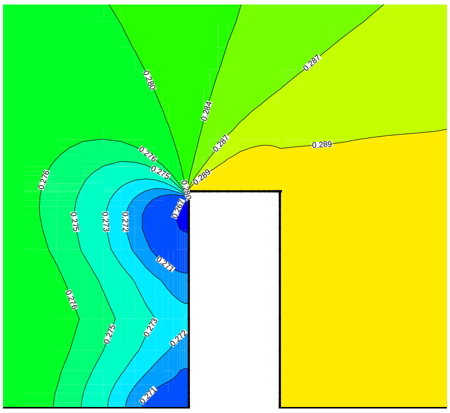


Fig 7: The computed pressure distribution close to the building

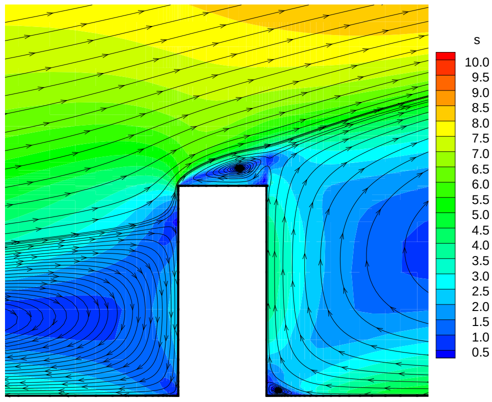


Fig 8: The streamtraces and the absolute velocity s

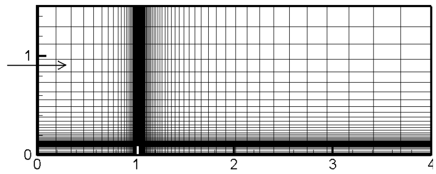


Fig 9: The computational grid

6 Conclusion

A numerical method for solution of 2D turbulent incompressible flow described by Navier-Stokes equations completed by two equations of k and ω was developed. Obtained numerical results correspond with theoretical and experimental data.

Acknowledgment

The financial support for the present project is partly provided by the Research Plan MSM No.6840770003 and the project of the Grant Agency of the Czech Republic No. 205/06/0727.

Reference

[1] Vierendeeles J., Riemslagh K., Dick E.: A Multigrid Semi-implicit Line-method for Viscous Incompressible and Low-Mach-Number Flows on High Aspect Ratio Grids,*Journal of Computational Physic 154, p.310-314, 1999*

[2] Wilcox D.C. Turbulence Modeling for CFD, *DCW Industries*

[3] Kopáček T.: Numerické řešení vazkého nestlačitelného proudění,*Diploma theses (in czech) ČVUT, 2007*

[4] Jirsák M., Zachoval D., Kopáček T.: Wind Tunnel Modelling of Ventilation and Infiltration Boundary Condition, *ARTI in Prague, 2006*

[5] Fuka V., Brechler J.: 2D model interakce proudění s pevnými překážkami. *Colloqium FLUID DYNAMICS 2006 (Jonas, Uruba, eds.), Inst. of Thermomechanics, Acad. of Sci., Prague, 2006,35–38.*

[6] Bodnár T., Fraunié Ph., Kozel K., Sládek I.: Numerical simulation of Comlex Atmospheric Boundary Layer Problems,*ERCOFTAC Buletin No.60, March 2004, p.5-12*

[7] Gulíková E., Bodnár T., Píša V.: Improvement of Numerical Models for Solution of Dust Air Pollution *Topical Problem of Fluid Mechanics 2006. Prague IT AV ČR, 2006, p. 63-66. ISBN 80-85918-98-6.*