23rd International Conference
ENGINEERING MECHANICS 2017
Svratka, Czech Republic, 15 – 18 May 2017

ATMOSPHERIC GAS FLOW AROUND AND INSIDE HIGH HALLS SUPPORTED WITH THE SIMULATION OF THE FANS

M. Kyncl*, J. Pelant**

Abstract: *Here we work with the system of equations describing the non-stationary compressible turbulent multi-component flow in the gravitational field, and we focus on the numerical solution of these equations. The RANS equations are discretized with the use of the finite volume method. The exact solution of the modified Riemann problem (original results) is used at the boundary faces. The presented computational results are computed with the own-developed code (C, FORTRAN, multiprocessor, unstructured meshes in general).*

Keywords: Atmospheric Gas Flow, RANS, Riemann Problem, Software, Fan Simulation, 3D.

1. Introduction

The physical theory of the compressible fluid motion is based on the principles of conservation laws of mass, momentum, and energy. The mathematical equations describing these fundamental conservation laws form a system of partial differential equations. In this contribution we consider the Reynolds-Averaged Navier-Stokes equations with the k-omega model of turbulence, shown in Wilcox (1998), Kok (2000). We focus on the real gas flow in the gravitational field. Further we suggest a method for the simulation of the simple fan, using two compatible boundary conditions, used previously in Kyncl et al. (2013). We focus on the numerical solution of these equations, using own-developed software.

2. Methods

For the discretization of the system we use either explicit or implicit finite volume method in order to discretize the analytical problem, represented by the system of equations in generalized (integral) form. In order to apply this method we split the area of the interest into the elements, and we construct a piecewise constant solution in time, as described in Feistauer et al. (2003). The crucial problem of this method lies in the evaluation of the so-called fluxes (and its Jacobians) through the edges/faces of the particular elements. We use the exact Riemann solver for the solution of the local problem at each face, with the theory shown in Toro (1997). At the boundary faces it is necessary to solve the incomplete Riemann problem, where the right-hand side initial condition is not known. It can be shown, that this right-hand side initial condition for the local problem can be partially replaced by the suitable complementary condition. Various original modifications of the Riemann problem (and exact solutions of these modifications) were shown and analyzed in Pelant (1996-2000), Kyncl (2011).

In this paper we simulate the fan using two connected boundaries. At these boundaries we solve the conservation laws, using the modification of the Riemann problem by the preference of the total quantities at the inlet, and the modification by the preference of the mass flow at the outlet. Using such boundary conditions it is necessary to compute the solution of the resulting non-linear problems. The combination of these two boundary conditions was used previously for the simulation of the propeller disk in Kyncl and Pelant (2013). The partial boundary condition with the preference of mass flow is sometimes being implemented with the use of some iterative process, guessing the correct values (for the pressure, density, velocity) in order to match the given mass flow through the boundary. In our approach we try to be as exact as possible, using our own original procedures. We follow the exact solution of the

* RNDr. Martin Kyncl, PhD.: Výzkumný a zkušební letecký ústav, a. s., Beranových 130, 199 05 Prague; CZ, kyncl@vzlu.cz
** RNDr. Jaroslav Pelant, CSc.: Výzkumný a zkušební letecký ústav, a. s., Beranových 130, 199 05 Prague; CZ, pelant@vzlu.cz

initial-value problem for the system of hyperbolic partial differential equations, the original analysis was shown in Kyncl et al. (2016).

The own-developed software (C, FORTRAN) is based on the finite volume method with the implicit or explicit time discretization, solution is computed on unstructured 3D meshes in general, MPI parallelizations (OpenMPI, MPICH, CUDA) are used, see also Kyncl et al. (2012). The large linear systems within the implicit method are solved with the implemented preconditioned GMRES matrix solver.

3. Examples

Here we show the computational results for the flow around and inside the high hall in the isothermal atmosphere. The initial condition was constant in the whole domain with the total temperature T_0 = 293.15K, zero velocity, and the total pressure distribution modified in the gravity field as $p_0 = p_{00} e^{9.81y/(RT_0)}$, where $p_{00} = 101325\,Pa$, y denotes the vertical coordinate, and R is the gas constant $R = 287.04\,Jkg^{-1}K^{-1}$. The horizontal fan simulation consisted of two connected boundary conditions: the inlet with the total temperature T_{0X} and the total pressure p_{0X}, which are the local values of the total temperature T_0 and the total pressure p_0, augmented by the influence of the fan

$$T_{0X} = T_0\left(1 + \frac{\gamma-1}{(2\gamma RT_0)} v_{FAN}^2\right),\ p_{0X} = p_0\left(1 + \frac{\gamma-1}{(2\gamma RT_0)} v_{FAN}^2\right)^{\frac{\gamma}{\gamma-1}}, \qquad (1)$$

with $\gamma = 1.4$, and $v_{FAN} = 5\,ms^{-1}$ is the fan parameter. Then the computed massflow was used to complement the incomplete the Riemann problem at the outlet boundary condition. Both direction of the flow through the fan were simulated. The geometry, the computational mesh (anisotropically refined), and the resulting isolines of velocity magnitude, density are shown in the Figs. 1 – 5.

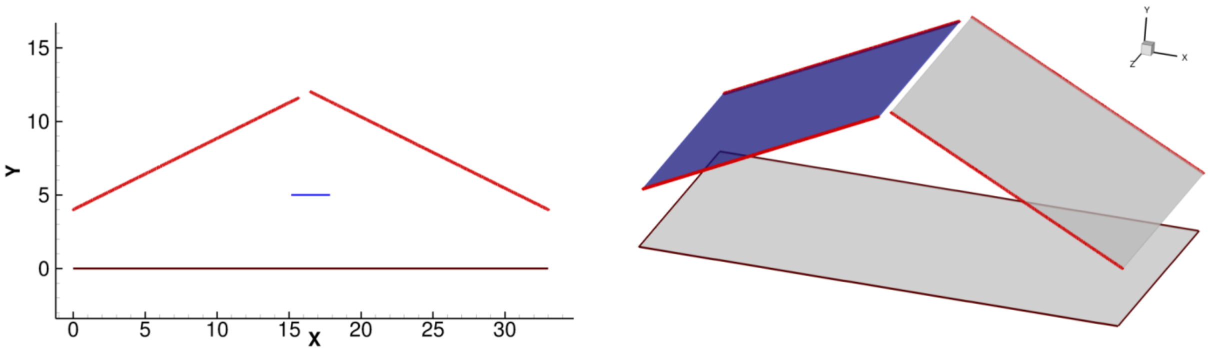

Fig. 1: Geometry used for the simulation. The fan is placed inside the high hall.

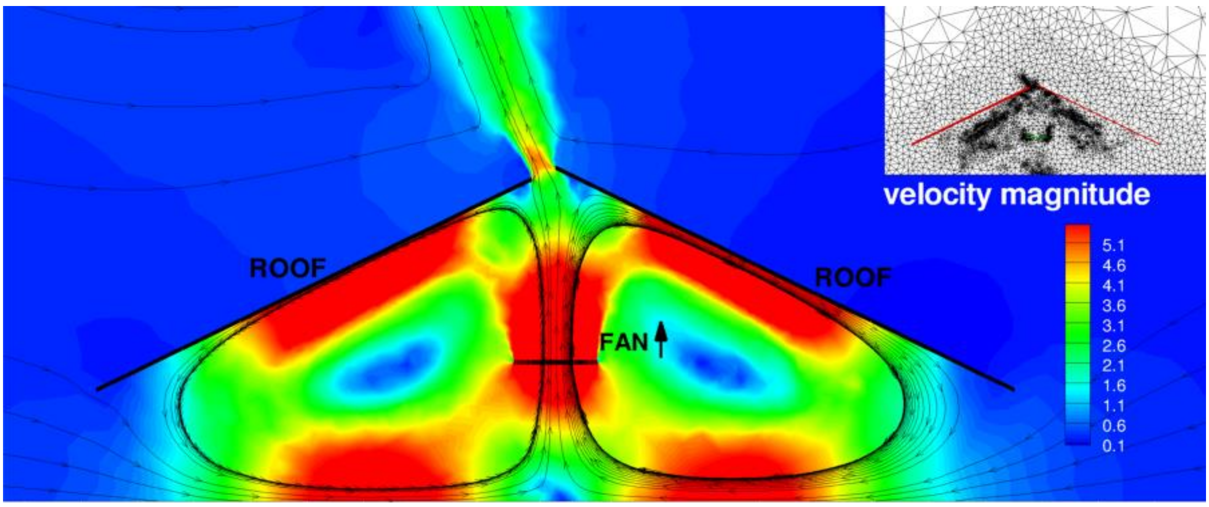

Fig. 2: The fan inside hall simulation, velocity magnitude isolines.
The direction of the flow is demonstrated by the streamtraces.

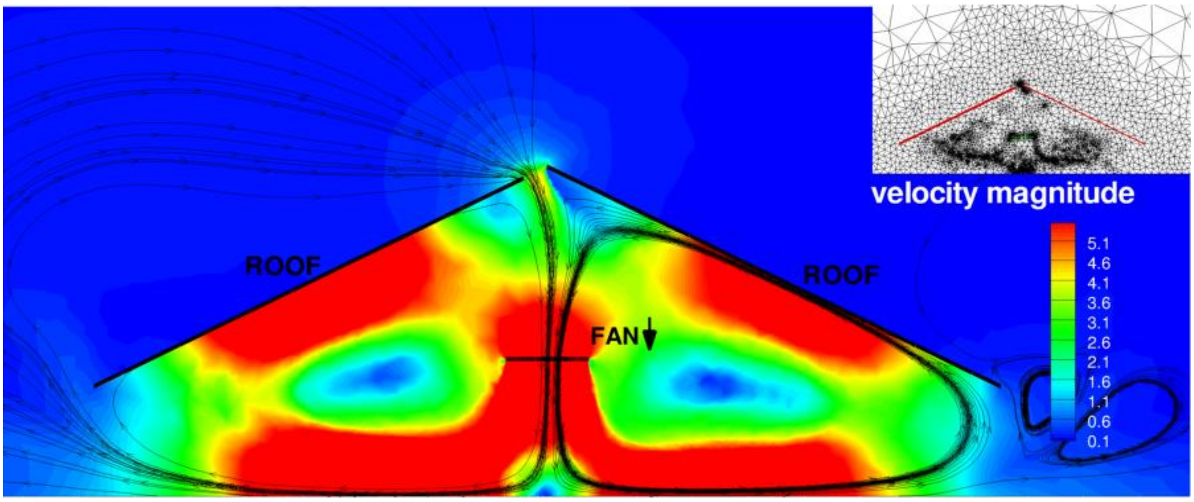

Fig. 3: The fan inside hall simulation, velocity magnitude isolines.
The direction of the flow is demonstrated by the streamtraces.

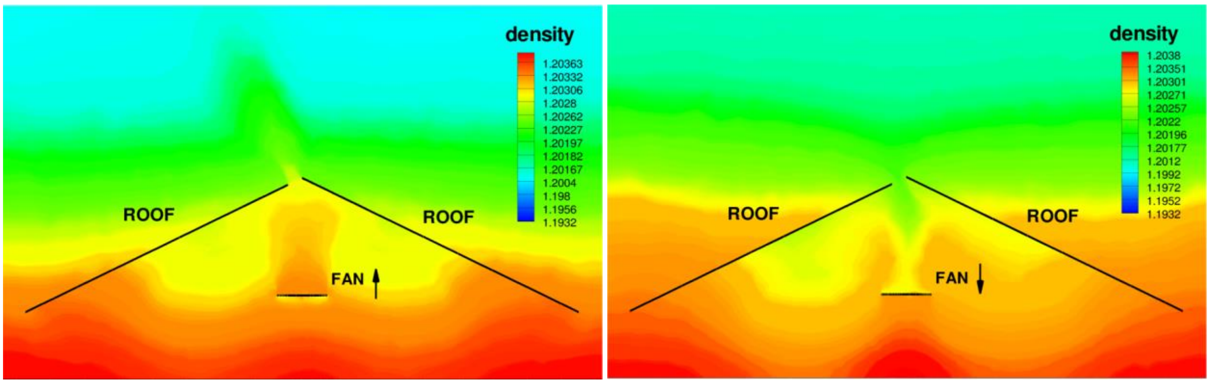

Fig. 4: The fan inside hall simulation, density isolines, comparison of two possible fan orientations. There is a visible effect of the gravitational force.

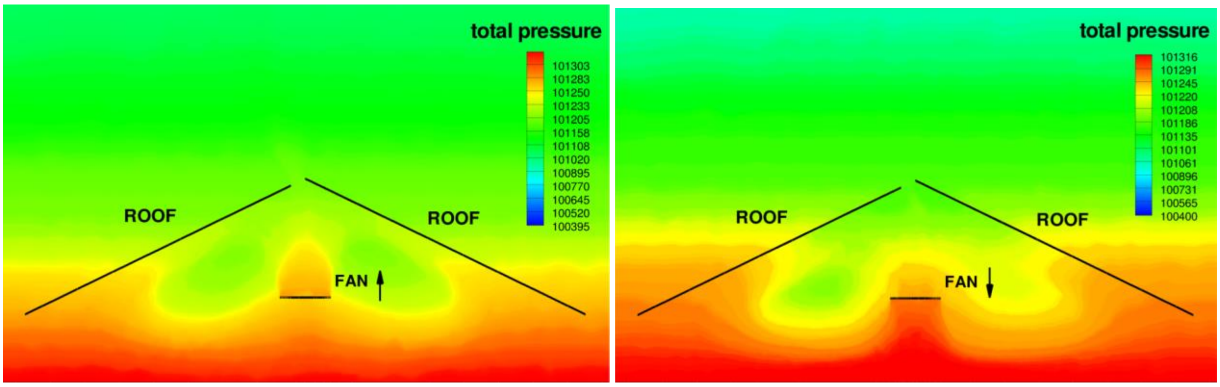

Fig. 5: The fan inside hall simulation, total pressure isolines, comparison of two possible fan orientations.

4. Conclusions

This paper shows the numerical simulation of the fan situated inside the high hall. The partial differential equations describing the conservation laws are solved numerically with the finite volume method. Own software was programmed. The modification of the Riemann problem and its solution was used at the boundaries. The combination of these boundary conditions was used for the simulation of the fan, which is the original result of this work.

Acknowledgement

The results originated with the support of Ministry of the Interior of the Czech Republic, project SCENT, Grant MSM 0001066902 of the Ministry of Education of the Czech Republic, and Ministry of Industry and Trade of the Czech Republic for the long-term strategic development of the research organization. The authors acknowledge this support.

References

Feistauer, M., Felcman, J., and Straškraba, I. (2003) Mathematical and Computational Methods for Compressible Flow, Oxford, England.

Kok, C.J. (2000) Resolving the dependence on free-stream values for k-omega turbulence model, in: AIAA Journal, Vol. 38, No. 7.

Kyncl, M. and Pelant, J. (2012) Implicit method for the 3d RANS equations with the k-w (Kok) Turbulent Model, Technical report R-5453, VZLU, Beranovych 130, Prague.

Kyncl, M. (2011) Numerical solution of the three-dimensional compressible flow, Doctoral Thesis, Prague.

Kyncl, M. and Pelant, J. (2013) Simulation of the Propeller Disk Inside the Symmetrical Channel, EPJ Web of Conferences,Volume 67, 2014, EFM13.

Kyncl, M. and Pelant, J. (2016) The Inlet and Outlet Boundary Problem with the Preference of Mass Flow, ECCOMAS Congress 2016.

Pelant, J. (1996-2000) Arti reports VZLÚ, z-65, z-67 to z-73. Prague.

Toro, E.F. (1997) Riemann Solvers and Numerical Methods for Fluid Dynamics, Berlin, Germany.

Wilcox, D.C. (1998) Turbulence Modeling for CFD, California, USA.