NUMERICAL SIMULATION OF TURBULENT FLOWS AROUND SINUSOIDAL HILLS

Luboš Pirkl and **Tomáš Bodnár**

Czech Technical University of Prague, Faculty of Mechanical Engineering, Department of Technical Mathematics

Introduction

Prediction of wind field over terrain plays an important role in many engineering applications. A numerical investigation of the flow over sinusoidal hill is presented in this work. We assume two-dimensional, steady, turbulent flow of incompressible fluid. We have computed four test cases with different geometries. The model is set-up for the wind flow according to Kim [1]. The introduced test cases were experimentally measured [1] and comparison with experiment and with other numerical simulations was made. The main characteristic of this type of test cases is the separation zone, that appears behind the hill.
The numerical method we have used is based on solving Reynolds Averaged Navier-Stokes equations with help of Finite Volume Method. We have implemented two models of turbulence. Mixing Length Model (T.B.) and Spalart-Allmaras turbulence model (L.P.) was used. As a numerical scheme we have used MacCormack explicit scheme.

Computational domain

Hill geometry

According to data published in [1] we have chosen 2D domain with four different sinusoidal single-hill terrain profiles having the following parameters:

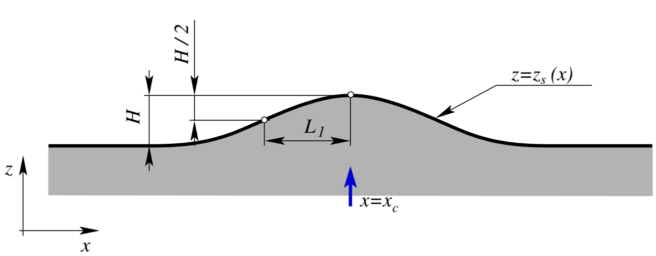

Figure 1: Hill geometry

Hill	**slope**	**height** H	**length** L_1
S3H4	0.3	4 cm	6.67 cm
S3H7	0.3	7 cm	11.67 cm
S5H4	0.5	4 cm	4.0 cm
S5H7	0.5	7 cm	7.0 cm

Table 1: Hill setup

The notation we use here to distinguish between hills with different slopes and heights is the same as in [1]. It means the SxHy stands for the hill with maximum slope $0.x$ and height $y\ cm$. The height of the whole computational domain is $0.5m$ and the length is $2.0m$.

Boundary conditions

For all the test cases we have used following boundary conditions:

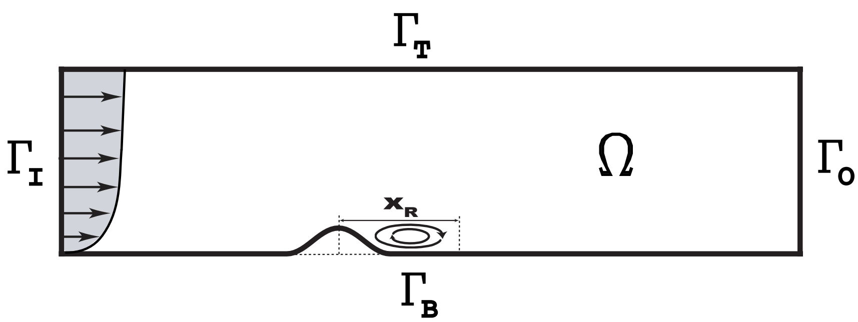

Figure 2: Computational domain

Γ_I ... Logarithmic velocity profile $u(y) = \frac{u_t}{\kappa} ln \frac{y}{y_0}$, $u_t = 0.33 m/s$, $y_0 = 0.0005 mm$ for $y < 0.25m$ and $u(y) = 7.0 m/s$ for $y > 0.25m$.
Γ_B ... $u, v = 0.0$,
Γ_T ... $\frac{\partial \vec{u}}{\partial \vec{n}} = 0.0$,
Γ_O ... $p = const.$

the rest of variables are extrapolated from the inside of the computational domain.

Numerical method

Governing system

Governing system are Reynolds Averaged Navier-Stokes equations:

$$\frac{\partial \bar{v}_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial \bar{v}_j}{\partial t} + \bar{v}_i \frac{\partial \bar{v}_j}{\partial x_i} = -\frac{1}{\rho} \frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_i} \Big(\nu \frac{\partial \bar{v}_i}{\partial x_j} + \tau_{ij}^R \Big) \tag{2}$$

Turbulence models

For eddy viscosity computation was used **Spalart-Allmaras** one-equational turbulence model. One differential equation is then added to system:

$$\frac{\partial \tilde{\nu}}{\partial t} + u_j \frac{\partial \tilde{\nu}}{\partial x_j} = C_{b1}[1 - f_{t2}]\tilde{S}\tilde{\nu} + \frac{1}{\sigma}\{\frac{\partial}{\partial x_j}[(\nu + \tilde{\nu})\frac{\partial \tilde{\nu}}{\partial x_j}] + C_{b2}\frac{\partial \tilde{\nu}}{\partial x_j}\frac{\partial \tilde{\nu}}{\partial x_j}\} - \left[C_{w1} f_w - \frac{C_{b1}}{\kappa^2} f_{t2}\right] \left(\frac{\tilde{\nu}}{d}\right)^2 + f_{t1} \Delta U^2 \tag{3}$$

The turbulent eddy viscosity is then:

$$\nu_T = \tilde{\nu} f_{v1}, \quad f_{v1} = \frac{\chi^3}{\chi^3 + C_{v1}^3}, \quad \chi = \frac{\tilde{\nu}}{\nu} \tag{4}$$

where $\chi = Re_T$ is Reynolds turbulent number. Model is completed by following formulas:

$$\tilde{S} \equiv |\Omega| + \frac{\tilde{\nu}}{\kappa^2 d^2} f_{v2}, \quad f_{v2} = 1 - \frac{\chi}{1 + \chi f_{v1}} \tag{5}$$

$$f_w = g \left[\frac{1 + C_{w3}^6}{g^6 + C_{w3}^6} \right]^{1/6}, \quad g = r + C_{w2}(r^6 - r), \quad r \equiv \frac{\tilde{\nu}}{\tilde{S} \kappa^2 d^2} \tag{6}$$

$$f_{t1} = C_{t1} g_t \exp\left(-C_{t2} \frac{\omega_t^2}{\Delta U^2} [d^2 + g_t^2 d_t^2] \right) \tag{7}$$

$$f_{t2} = C_{t3} \exp(-C_{t4} \chi^2) \tag{8}$$

The following table gives the model constants present in the formulas above:

σ	C_{b1}	C_{b2}	κ	C_{w1}	C_{w2}	C_{w3}	C_{v1}	C_{t1}	C_{t2}	C_{t3}	C_{t4}
$\frac{2}{3}$	0.1355	0.622	0.41	$C_{b1}/\kappa^2 + (1 + C_{b2})/\sigma$	0.3	2.0	7.1	1.0	2.0	1.1	2.0

The second turbulence model is **Mixing Length Model**. This model belongs to the group of algebraic models and is presented e.g. in [3].

Numerical scheme

For computation The MacCormack explicit scheme was used:

$$\mathsf{W}_{i,j}^{n+\frac{1}{2}} = \mathsf{W}_{i,j}^n - \frac{\Delta t}{|\mathsf{D}_{i,j}|} \sum_{k=1}^{4} \{ (\mathsf{F}_k^n - \mathsf{R}_k^n) \Delta y_k - (\mathsf{G}_k^n - \mathsf{S}_k^n) \Delta x_k \} \tag{9}$$

$$\left(\mathsf{W}_{i,j}^{n+1} \right) = \frac{1}{2} \left(\mathsf{W}_{i,j}^n + \mathsf{W}_{i,j}^{n+\frac{1}{2}} - \frac{\Delta t}{|\mathsf{D}_{i,j}|} \sum_{k=1}^{4} \{ (\mathsf{F}_k^{n+\frac{1}{2}} - \mathsf{R}_k^{n+\frac{1}{2}}) \Delta y_k - (\mathsf{G}_k^{n+\frac{1}{2}} - \mathsf{S}_k^{n+\frac{1}{2}}) \Delta x_k \} \right) \tag{10}$$

$$\mathsf{W}_{i,j}^{n+1} = \left(\mathsf{W}_{i,j}^{n+1} \right) + \mathsf{DW}_{i,j}^n \tag{11}$$

where artificial viscosity DW can be expressed:

$$\mathsf{DW}_i^n = \epsilon_2 \Delta x^3 \frac{d}{dx} |\mathsf{W}_x| \mathsf{W}_x \Big|_i^n + \epsilon_4 \Delta x^4 \mathsf{W}_{xxxx} \Big|_i^n \tag{12}$$

after replacing derivations we obtain:

$$\mathsf{DW}_i^n = \epsilon_2 \left[|\mathsf{W}_{i+1}^n - \mathsf{W}_i^n| (\mathsf{W}_{i+1}^n - \mathsf{W}_i^n) - |\mathsf{W}_i^n - \mathsf{W}_{i-1}^n| (\mathsf{W}_i^n - \mathsf{W}_{i-1}^n) \right] + \epsilon_4 (\mathsf{W}_{i-2}^n - 4\mathsf{W}_{i-1}^n + 6\mathsf{W}_i^n - 4\mathsf{W}_{i+1}^n + \mathsf{W}_{i+2}^n) \tag{13}$$

Results

$S3Hx$ Test cases

For these $S3Hx$ (gentle-sloped) test cases we all (both models) obtained small separation zones near the wall, but mentioned experiment ([1]) obtained none. The following figures show results gained with MLM model.

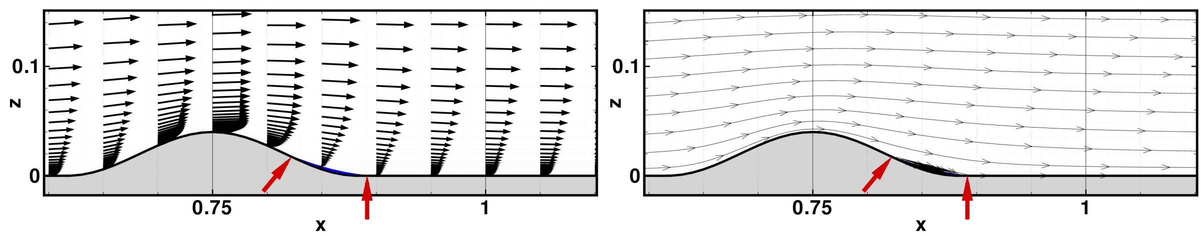

Figure 3: Flow over S3H4 hill. Separation and reattachment points marked by arrows.

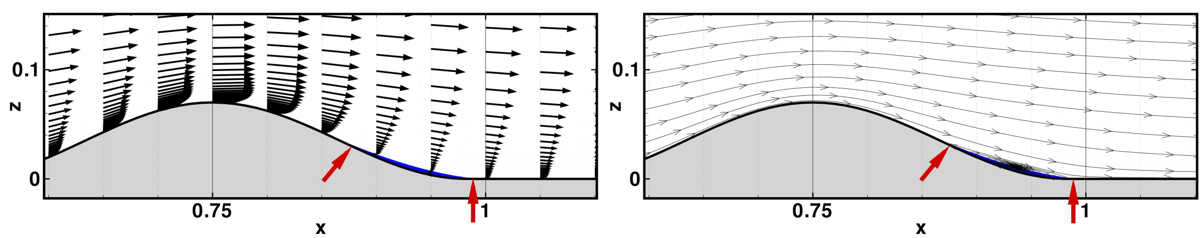

Figure 4: Flow over S3H7 hill. Separation and reattachment points marked by arrows.

$S5Hx$ **Test cases**

On $S5Hx$ (sharp-sloped) test cases were obtained following results. The following figures show results gained with S-A model.

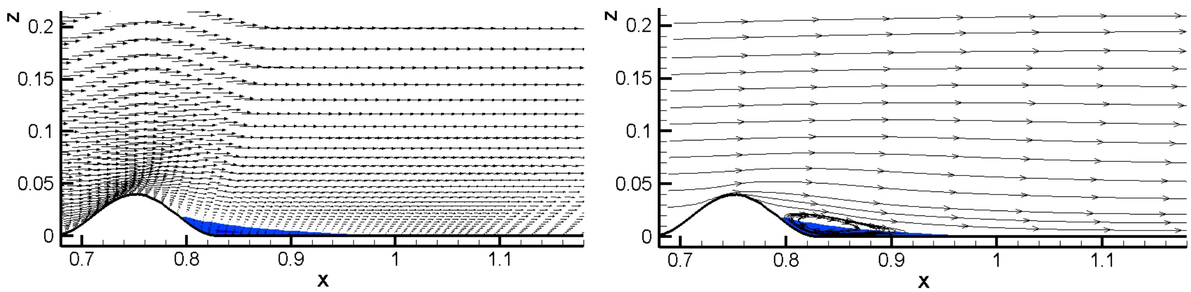

Figure 5: Flow over S5H4 hill.

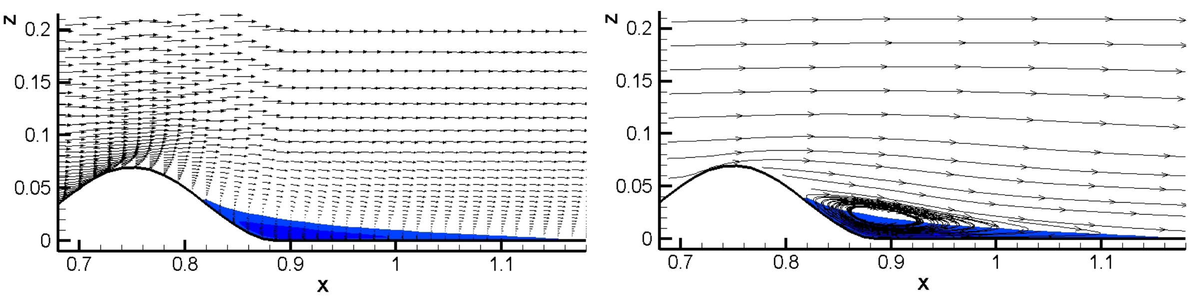

Figure 6: Flow over S5H7 hill.

The blue color signs negative component of u (velocity in x-direction) and signalize the separation zone.

The comparison of $S5Hx$ (sharp-sloped) test cases is shown in following table:

Hill	**Experiment**	**Standard** $k-\epsilon$	**RNG** $k-\epsilon$	**Low-Re model**	**MLM**	**S-A**
S5H4	**5.25 ± 0.5**	2.5	3.42	4.55	**5.0**	**5.8**
S5H7	**4.30 ± 0.3**	-	3.97	4.42	**4.7**	**5.7**

Table 2: Reattachment point position (x_R/H)

Results from first four columns are presented in [1], the last two columns are results from both authors.

Conclusion

Presented models appear to be applicable. The differences in obtained results may be caused, by various reasons. Each model itself has its advantages and limitations. To catch separation zones are recommended more sophisticated models e.g. EARSM models.
It is also necessary to be careful of choice of the computational mesh. Presented test cases require very fine mesh especially at the wall, to catch great velocity gradients. In [1] author even shows that orthogonal mesh gives different results than the non-orthogonal one.
Overall, presented results confirm that the theoretical model is applicable for the prediction of velocity field and flow characteristics.

Acknowledgement

The financial support for the present project was partly provided by the *Czech Grant Agency* under the *Grant No.205/06/0727* and by the Research Plan *MSM 6840770010* of the *Ministry of Education of Czech Republic*.

References

[1] Kim, H. G., Lee, C. M., Lim, H. C., and Kyong, N. H.: An experimental and numerical study on the flow over two-dimensional hills. Journal of Wind Engineering and Industrial Aerodynamics, vol. 66, no. 1:(1997) pp. 17-33.

[2] Dvořák R., Kozel K.: Matematické modelovaní v aerodynamice. Vydavatelství ČVUT, 1996.

[3] Bodnár T., Numerical Simulation of Flows and Pollution Dispersion in Atmospheric Boundary Layer. CTU Thesis, 2003

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

[5] Příhoda J., Louda P., Matematické modelování turbulentního prodění,Vydavatelství ČVUT, 2007.

[6] Pirkl L., Numerical Simulation of Incompressible Flows with Variable Viscosity, Diploma Thesis, CVUT FS MMT , Prague, 2007.

[7] Spalart P.R., Allmaras S.R., A One-equational Turbulence Model for Aerodynamic Flows, AIAA, Paper 92-0439, (1992).