30th International Conference
ENGINEERING MECHANICS 2024
Milovy, Czech Republic, May 14 – 16, 2024

LES SIMULATIONS OF AIRFLOW AROUND RECTANGLE WITH SIDE RATIO 2:1 AND THEIR COMPARISION WITH EXPERIMENTS

Ledvinková B.[*], Hračov S.[**], Macháček M.[***]

Abstract: *Our contribution is focused on the comparison of the experimental investigations and of the numerical 3D LES simulations of the airflow around sharply edged rectangle with side ratio 2:1. The rectangle object was exposed to the airflow having a given velocity at different angles of the wind attack in the wind tunnel with the aim to obtain the curves of the aerodynamic coefficients and Strouhal number depending on the impact angle. The comparative numerical 3D simulations of the wind tunnel testing were performed using COMSOL Multiphysics and OpenFoam both incorporating the Large Eddy Simulation (LES) method.*

Keywords: Rectangle 2:1, wind tunnel, LES simulation, aerodynamic characteristics.

1. Introduction

The flow around U-profiles with the constant side ratio 2 differing in the inner depth of their vertical sides and the effect of the porosity or their flanges at different attack angles were studied in the wind tunnel by Hračov and Macháček (2020). Ledvinková et al. (2021) and Ledvinková et al. (2022) with the aim to compare simulated results with the experimental ones performed the 2D URANS simulations with the same geometry and conditions. The qualitative compliance between the simulated and experimental aerodynamic mean coefficients was achieved, however 2D URANS simulations are not capable to describe realistically the flow characteristics.

In this work we thus proceed to the 3D modelling and as the first verifying study the turbulent air flow around the bluff body having rectangular cross section with side ratio 2:1 using Large Eddy Simulation method is modelled. Our aim is to evaluate aerodynamics coefficients, to investigate the flow characteristics for different attack angles and to compare computed results with the experimental ones.

2. Computational settings

Large Eddy Simulation method (LES) solves explicitly large eddies whereas the effect of small eddies is accounted via the sub-grid scale model. The implementations of the LES method with standard Smagorinsky sub-grid model in Comsol Multiphysics simulation platform based on the Finite Element Method (FEM) and OpenFoam open source code using the finite volume method were used.

The investigated rectangle body (30 x 15cm) was placed into the larger square computational domain 7.5 x 7.5 m as apparent from Fig. 1. The size of the computational domain was chosen in such a way so that the blockage effect is negligible. Due to the time demanding computations for the inlet flow velocity 14m/s used in the wind tunnel experiments (Hračov and Macháček, 2020), the velocity of the inlet flow was chosen to be 2.8 m/s (corresponding to the Reynold's number Re = 2.7e4) for the most of performed

[*] Ing. Blanka Ledvinková, PhD.: Institute of Theoretical and Applied Mechanics of the Czech Academy of Sciences, Prosecká 809/76; 190 00, Prague; CZ, ledvinkova@itam.cas.cz

[**] Ing. Stanislav Hračov, PhD.: Institute of Theoretical and Applied Mechanics of the Czech Academy of Sciences, Prosecká 809/76; 190 00, Prague; CZ, hracov@itam.cas.cz

[***] Ing. Michael Macháček, PhD.: Institute of Theoretical and Applied Mechanics of the Czech Academy of Sciences, Prosecká 809/76; 190 00, Prague; CZ, machacek@itam.cas.cz

doi: 10.21495/em2024-182

simulations. Periodic boundary conditions were imposed on side and lower-upper surfaces (cf. Fig. 1). The spanwise length of the computational domain was set to be 0.3 m. This value equals to the along wind dimension of the rectangle object and fulfils the requirement for the spanwise length suggested by Tamura et al. (1998).

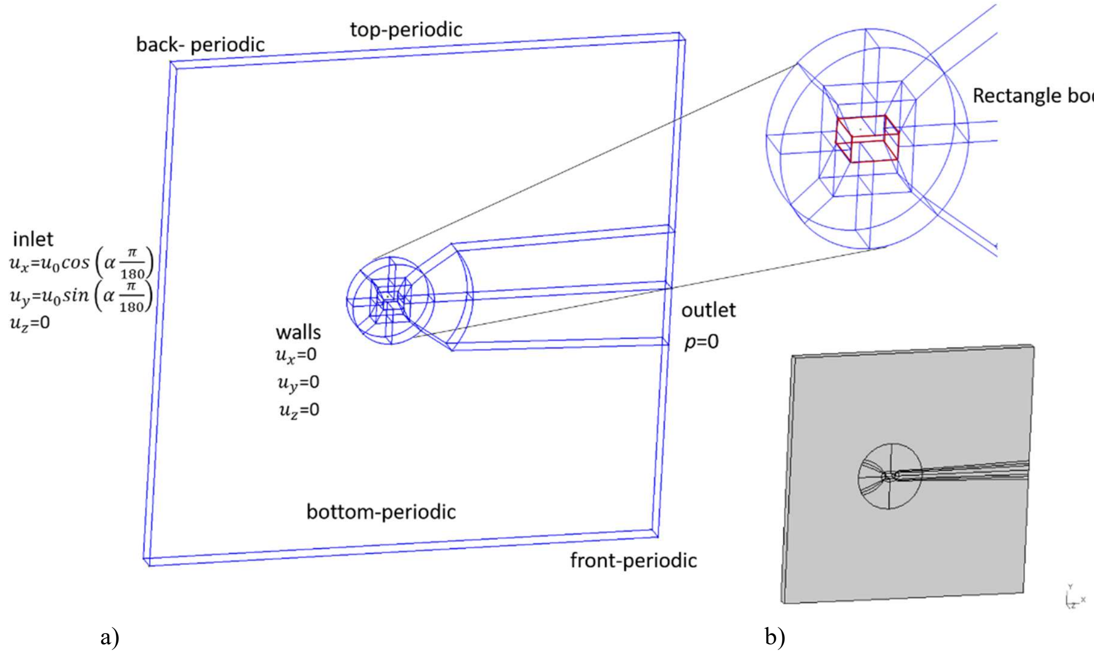

Fig. 1: a) Computational domain for OpenFoam calculations with initial and boundary conditions, b) Computational domain for Comsol Multiphysics calculations.

The computational meshes are not identical for Comsol Multiphysics and OpenFoam computations (due to the different meshing software and various requirements of FEM/FVM method) but they have similar structure. The rectangle body is surrounded by the structured boundary layer having 6 cell layers, the width of the first cell layer is 1.0e-4 m and the expansion ratio is 1.2 in the x-y plane. The domains in the circle surrounding of the rectangle object and in the wake area are also covered by structured mesh, the cell size increases continuously with the increasing distance from the body as shown in Fig. 2. The rest of the computational domain is covered with unstructured mesh, cf. Fig. 2. The described 2D mesh was extruded along the z-dimension by 24 uniformly distributed cells. The mesh for Comsol calculations consists of 900 000 elements, while the total cell number for OpenFoam calculations is about 3.5 million.

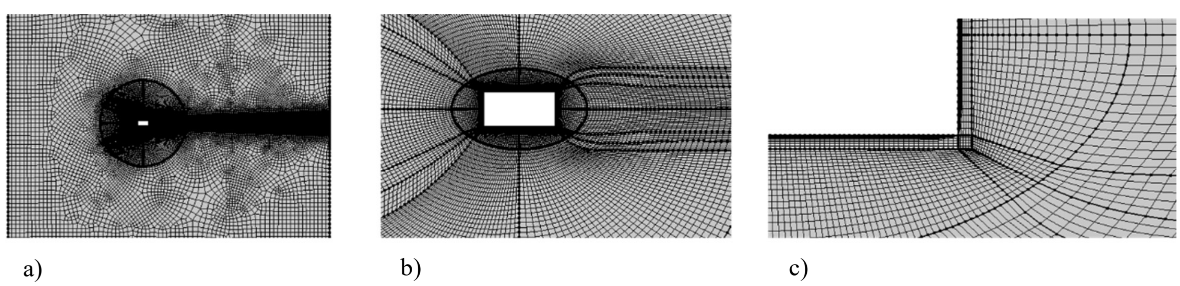

Fig. 2: Computational mesh for Comsol calculations- projection into x-y plane –a) whole domain, b) proximity of the rectangle body, c) detail of the corner of the rectangle.

For the OpenFoam calculations, the second-order centred differentiation scheme is used for diffusive terms and convective terms are approximated by Linear Upwind Stabilised Transport scheme (LUST). Time progress is managed by the backward differentiation formula (BDF) approach (inspired by Bruno et al., 2010). The PIMPLE algorithm with the automatic time step regulation was used, maximum Courant number was set to be 2.4.

Calculations in Comsol Multiphysics software are performed by the Algebraic Multigrid method using GMRES solver and generalized α method for the time discretization. In order to achieve reasonable computational time P1+P1 fluid discretization assuming linear elements for both pressure and velocity field.

3. Results

The values of the drag and the lift forces were evaluated by the integration of the *x*-and *y* components of the total stress force over all the walls of the investigated body for several angles of wind attack α. The reference dimension for the calculation of coefficients is across wind dimension 0.15 m.

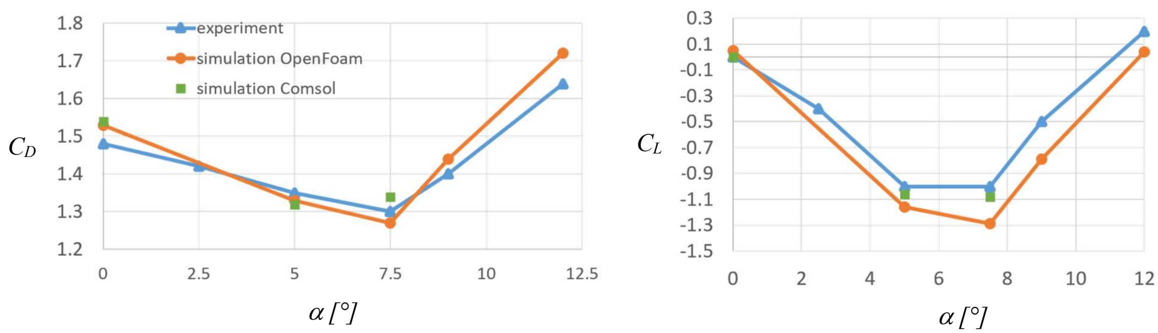

Fig. 3: Mean drag and lift coefficient for the rectangle with side ratio 2 depending on the impact angle - comparison of the results obtained from the simulations performed in Comsol Multiphysics and OpenFoam and from the experiments in the wind tunnel.

The comparison of the calculated and experimental mean aerodynamic coefficients for several positive values of the angle of attack is shown in Fig. 3. As apparent, the simulated and measured values show qualitatively same trends and the position of the minimum corresponds approximately to the same impact angle.

The values of Strouhal number related to the vortex shedding frequency for the zero angle of attack and two different wind velocities are listed in Tab. 1. Strouhal number seems to be only slightly velocity dependent, the values calculated by Comsol Multiphysics correspond better to the measured ones.

	v = 2.8 m/s	v = 14 m/s
experiment	0.087	0.092
simulation-Comsol Multiphysics	0.078	0.080
simulation- OpenFoam	0.068	not calculated yet

Tab. 1: Experimentally and numerically determined Strouhal number for the wind velocity v=2.8 m/s and v = 14 m/s and zero impact angle.

The experimental dependence of Strouhal number on the impact angle is shown in Fig. 4a. Its value fluctuates around 0.09 for small impact angles α < 5 °, in the proximity of the angle α = 5 ° the frequency spectrum is ambiguous and there is a step increase for angles α > 7 °. This rise was observed also in simulations. The calculated time dependencies of the lift coefficient and the values of Strouhal numbers for three values of impact angles are shown in Fig. 4b–d.

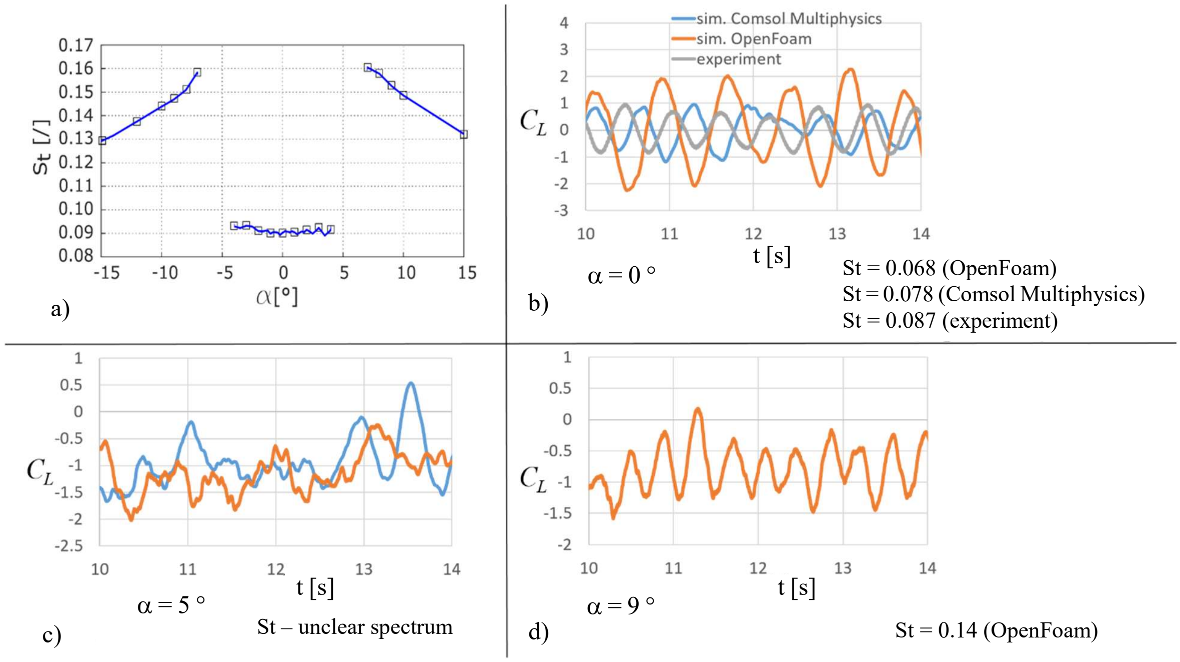

Fig. 4: a) Dependence of Strouhal number on impact angle obtained experimentally for v = 12.8 m/s, b–d) time dependence of lift coefficient and Strouhal number for various angles of attack obtained from OpenFoam and Comsol Multiphysics simulations.

4. Conclusions

The results of our simulations show qualitatively same trends for the mean aerodynamics coefficients and for Strouhal number depending on the impact angle as the experimental results obtained by the static measurement in the wind tunnel. However, the numerical results provided by Comsol Multiphysics simulations correspond better with experiments than those obtained from OpenFoam simulations.

Acknowledgement:

The kind support of Czech Scientific Foundation project No. 24-13061S is gratefully acknowledged.

Computational resources were provided by the e-INFRA CZ project (ID:90254), supported by the Ministry of Education, Youth and Sports of the Czech Republic.

The cooperation with Ing. Martin Kožíšek from HUMUSOFT s.r.o. is gratefully acknowledged.

References

Holmes, J. D. (2018) *Wind Loading Structures*, CRC Press, 3rd edition.

Hračov, S. and Macháček, M. (2020) Susceptibility of U-profiles with different geometry and porosity to galloping. *EASD Procedia EURODYN*, pp. 621–630.

Ledvinková, B., Hračov, S. and Macháček, M. (2021) Numerical study of the air flow around the U-profile. In: *Proc. of Computational Mechanics*, Srní, pp. 138–141.

Ledvinková, B., Hračov, S. and Macháček, M. (2022) Determination of aerodynamic coefficients for air flow around U-profiles with different flange porosities. In: *Proc. of the 28th Conference* Engineering *mechanics,* Milovy, pp. 237–240.

Bruno, L., Francos, D., Coste, N. and Bosco, A. (2010) 3D flow around a rectangular cylinder: A computational study. *J. Wind Eng. Ind. Aerodyn.*, 98, pp. 263–276.

Tamura, T., Miyagi, T. and Kitagishi, T. (1998) Numerical prediction of unsteady pressures on a square cylinder with various corner shapes. *J. Wind Eng. Ind. Aerodyn.*, 35, pp. 275–298.