10.1515/tvsb-2016-0006

Transactions of the VŠB – Technical University of Ostrava

Civil Engineering Series, Vol. 16, No. 1, 2016

paper #6

Vladimira MICHALCOVA[1], Lenka LAUSOVA[2]

NUMERICAL CALCULATION OF AERODYNAMIC ROUGHNESS OF CHIMNEY JACKETED WITH CORRUGATED SHEETS

Abstract

The article deals with the influence of a shape of smokestack casing on the total load from wind effects. It describes possibilities of defining an equivalent aerodynamic roughness and aerodynamic drag coefficient for numerical modelling of the flow around a circular cylinder. The aim of this study is to solve the force coefficient for the smokestack of a cylindrical shape, which is jacketed with shaped sheets. The flow around a smokestack is solved in the ANSYS Fluent software using the DES model.

Keywords

Circular cylinder, drag coefficient, aerodynamic roughness, CFD, DES model, high Reynolds number, boundary layer, wall function.

1 INTRODUCTION

The coating of smokestacks by a classic smooth plain metal sheet is a standard solution that entails certain complications. This is due to the dilatation of material and the combination of different materials for the shell support and the exterior coating (Fig.1). For these reasons, there is often a requirement that the smokestack were jacketing with a shaped metal sheet. This covering causes a problem particularly in defining the value of wind-drag coefficient, which is important for determination of total load of wind effects.

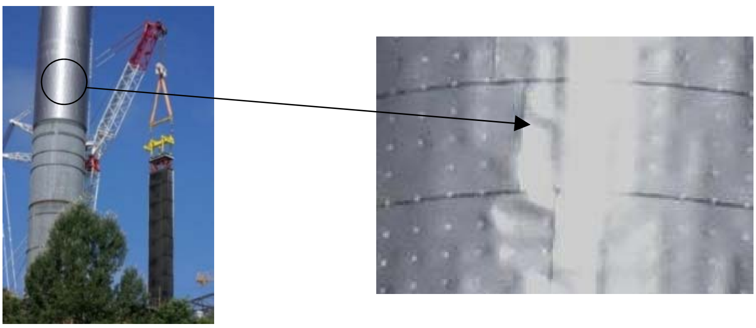

Fig.1: Deformation of sheathing of a smokestack

The calculation according to valid standard EN 1991-1-4 takes account only of the flow around cylinders with a rough surface, but does not specify what the equivalent surface roughness

[1] Ing. Vladimira Michalcova, Ph.D., Department of Structural Mechanics, Faculty of Civil Engineering, VŠB - Technical University Ostrava, Ludvíka Podéště 1875/17, 708 33 Ostrava-Poruba, Czech Republic, phone: (+420) 597 321 348, e-mail: vladimira.michalcova@vsb.cz.

[2] Ing. Lenka Lausova, Ph.D., Department of Building Structures, Faculty of Civil Engineering, VŠB - Technical University Ostrava, Ludvíka Podéště 1875/17, 708 33 Ostrava-Poruba, Czech Republic, phone: (+420) 597 321 326, e-mail: lenka.lausova@vsb.cz.

should be assigned to a cylinder with a smooth surface. Taking into consideration only the wave-height regardless of its shape according to the requirements of the applicable standards leads to a large increase of the drag coefficient value, in standards referred to as the coefficient forces $c_{f,0}$. The increase is often up to twofold.

The aim of this work is to find possibilities according to the CFD codes in the ANSYS Fluent software to determine the value of the equivalent aerodynamic roughness and then aerodynamic drag coefficient [1,2] of flow around a smokestack sheathed with two sheet metal textures (corrugated and trapezoidal), which gives a measure of loading of the structure from wind effects.

2 NUMERICAL MODELLING

The issue of the flow around a circular cylinder with high Reynolds numbers is a complicated phenomenon, whose solution is dealt with by many international teams, either experimentally [3,4] or numerically [5-8].

The value of the drag coefficient c_d depends on the Reynolds number and is very sensitive to the surface roughness of the flowed-around surface. The transition area decreases with increasing roughness Re and simultaneously increases the minimum value of c_d (Figure 2). This issue finds use also in other fields [9,10].

Within this work, flow around a real smokestack of a circle cross-section with a diameter of 3.36 m is simulated. The cladding consists of SP18/76 corrugated sheet from the firm Kovové profily, spol. s.r.o. and SAT158 trapezoidal sheet from the firm Satjam. Both have a wave height of 18 mm and are relatively similar in geometry. Basic wind speed is assumed at 20 m/s. Airflow to the value of Reynolds number $Re = 4.5 \cdot 10^6$ is far within the supercritical area and therefore in the boundary layer around the wall of the chimney it is possible to assume fully developed turbulence [11-14].

2.1 Flow modelling near the wall

During numerical solution a cooperation with the experimental research [15,16] is suitable. The flow near the wall can be modelled in two ways. The first one defines a wall function whereby the area of laminar sublayer and transition layer is spanned, i.e. the area between the wall and the area of a fully developed turbulent flow where molecular and turbulent viscosity are tangible. The second method consists of the detailed modelling of a real geometry including viscous sublayer in accordance with the meshing fineness. Modelling using the real geometry of a smokestack covering requires a large number of cells in the computational area, which is currently unrealistic to be solved by desktop PCs.

Thus the modelling of the flow around non-profiled cylinder with a significantly smaller number of cells remains an option. The influence of the actual shape of the casing is replaced by using the wall functions, which is a set of semi empirical formulas and functions applied in the first cell wall. The wall function comprises the wall law for the mean flow speed and formulas for turbulent quantities near the wall. The wall function is determined by a modified logarithmic law where the function of surface roughness of the flow around a surface with significant uneveness can be expressed by using aerodynamic roughness:

$$\Delta B = \frac{1}{\kappa} \cdot \ln f_r \tag{1}$$

where:

B – is an additive constant describing the function of roughness [-],

κ – von Karman constant [-] and

f_r – roughness function in Fluent.

A universal description of the roughness function f_r for different types of rough surfaces does not exist, but it is generally found to correlate well with dimensionless K_S^+:

$$K_S^+ = \frac{\rho \cdot K_S \cdot u^*}{\mu} \tag{2}$$

where:

K_S – is physical roughness height [m],

ρ – density of the flowing medium [kg·m^{-3}],

u^* – friction velocity [m·s^{-1}] and

μ – dynamic viscosity of the flowing medium [Pa·s].

The function of roughness of the flow around a surface with significant unevenness can be defined as:

$$\Delta B = \frac{1}{\kappa} \cdot \ln\left(1 + C_S \cdot K_S^+\right) \tag{3}$$

where:

C_S – is a roughness constant, depending on the type of roughness [-].
The Fluent manual recommends a value of 0.5 with options ranging from 0.5 to 1.1.

Aerodynamic roughness of the flow around the body is determined by two constants K_S and C_S, which enter into the calculation using wall functions within the boundary conditions. The computational grid must be designed so that the value of K_S reaches no further than the middle of the first cell by the wall.

Since the values of C_S and K_S for the simulated process are not known, the solution of the problem is divided into two phases. In the first phase the equivalent aerodynamic roughness of corrugated iron is determined. This is about determining the values of C_S and K_S based on the assessed loss of pressure and flow velocity in the task that allows a retrospective checking of the measured values which are traceable in the literature, e.g. [17]. In the second phase of the solution the constants C_s and K_s are used for the wall function when calculating the drag coefficient of the flow around the cylinder c_d.

2.1 Determining equivalent aerodynamic roughness

The task is modelled as a turbulent flow in a rough pipe with a diameter of one meter and a length of 20 meters. The geometry of the walls is identical to the geometries of two selected sheets. This is a 2D axisymmetric task of flow speed $v = 20$ m/s and 20 % turbulence intensity. The computational area is 20 meters long and it is divided into three parts in order to ensure a steady stream. The non-profiled ramp at the start is one meter long, the non-profiled ramp at the end is 4 meters long. Middle length of 15 m has geometry of the wall identical to the geometry of the sheet. Grids consist of approximately 160,000 tetra cells. Task is solved as a stationary problem using the SST-ω model.

The aim is to design constants C_S and K_S which allow to substitute the actual roughness of the flow around the body with wall functions. Pressure losses are monitored for a given velocity v = 20 m/s (Fig.2), and on the basis of their knowledge it is possible to determine the values of the constants. As seen in the figure, the pressure losses on the corrugated and trapezoidal sheet are nearly identical. It follows that the equivalent aerodynamic roughness of both sheets is identical.

On the basis of subsequent testing tasks constant values C_S and K_S are sought to define the equivalent aerodynamic roughness so that the pressure losses at a given velocity remain unchanged. One concern is to maintain the recommended value of $C_S = 0.5$.

In these testing tasks the airflow is simulated in a dimensionally identical conduit, with the difference that the 15-meter areas of profiled walls are replaced with straight walls. The roughness of the actual profiled surfaces is gradually replaced with the searched equivalent aerodynamic roughness defined by the above-mentioned constants C_S and K_S (1) to (3). The same pressure loss was achieved as in modelling of the actual geometry at relatively little roughness constant K_S = 1.0 mm. During calculation with the value of the roughness constant according to the standard so that it corresponded to the actual wave height, thus K_S = 18.0 mm, it came up to twice the friction (Fig.2). The roughness was overvalued by that.

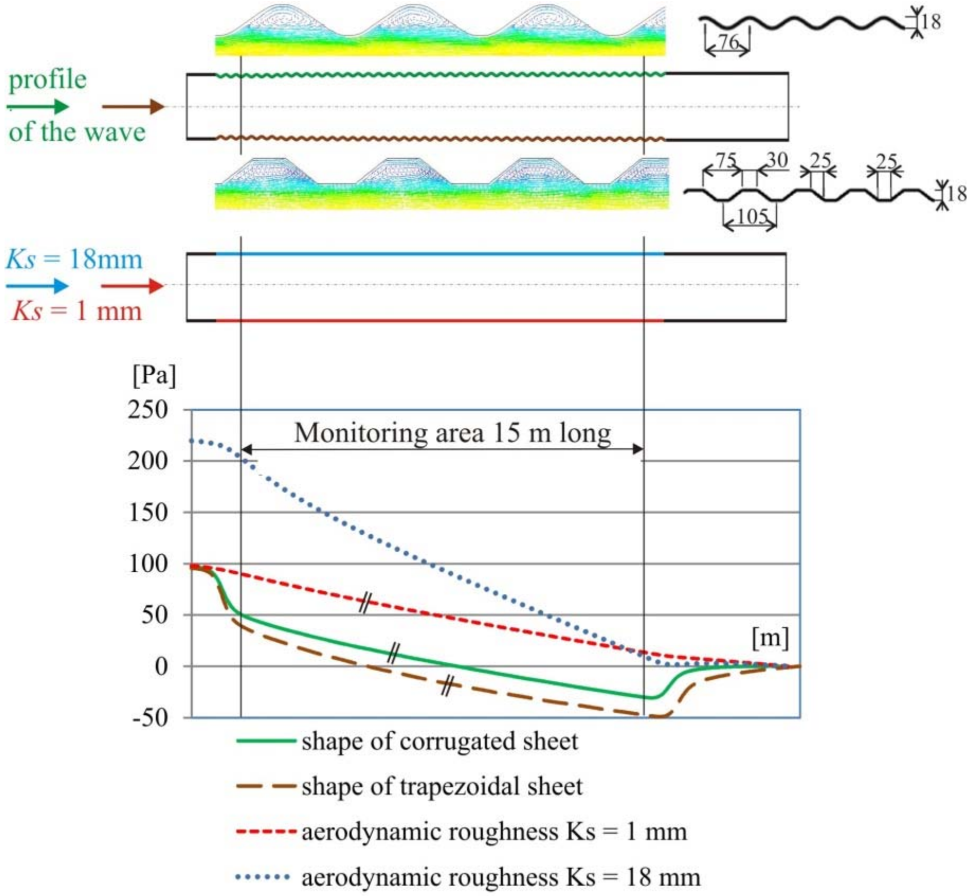

Fig.2: Pressure and pressure loss in the longitudinal axis of the pipe

Based on the results of this testing part of the task, the equivalent aerodynamic roughness for both shapes of the used sheets is defined:

- roughness constant, depends on the type of roughness: C_S = 0.5 [-],
- roughness constant, depends on the physical roughness height: K_S = 1.0 mm.

It can be assumed that the constants have been designed correctly and their values can be used in the second phase of the task - to define drag coefficient that is required to calculate the effects of wind loading on the structure.

2.2 Determining drag coefficient

This is a 3D task dealing with the flow around the cylinder (chimney). A flow around 5 m long section of a non-profiled smokestack in the computational area of dimensions 80 × 30 × 5 m (Fig.3) is modelled. The grid consists of 360,000 tetra cells. First cells at the wall are formed by a boundary layer and set in such a way that the height of the equivalent aerodynamic roughness does not reach further than the mid-height of the first cell at the wall. This is solved as a non-stationary problem with the use of DES model.

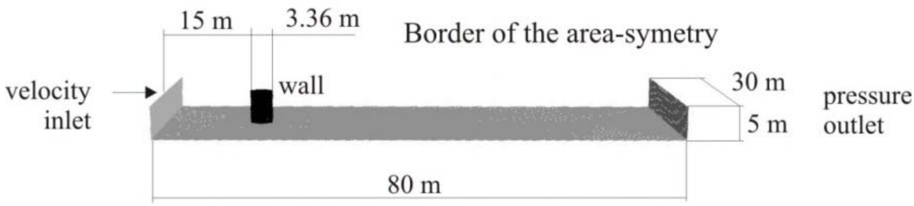

Fig.3: Scheme of the calculation area with boundary conditions

Two calculations are performed:

- A calculation using an equivalent aerodynamic roughness with the value of a roughness constant defined by the conclusions of the previous chapter $K_S = 1.0$ mm.
- A calculation using an aerodynamic roughness with the value of a roughness constant specified by the requirements of valid standards in which K_S corresponds with a wave height, namely $K_S = 18$ mm.

3 RESULTS

Vortex structures behind the flow around the cylinder are illustrated in Fig.4 and time records of sought-after drag coefficient c_d in Fig.5. The value of the drag coefficient c_d determined from the numerical simulations as well as calculated according to current standards is evident in Fig.6.

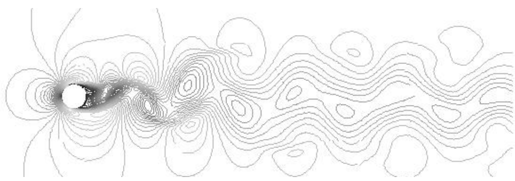

Fig.4: Vortices behind flow around smokestack

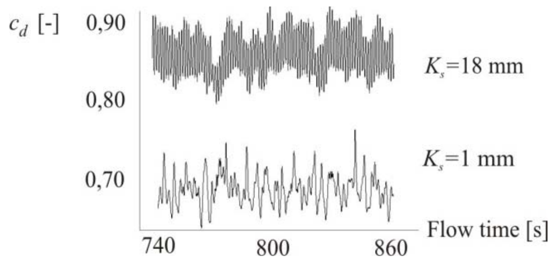

Fig.5: Drag coefficient in flow time

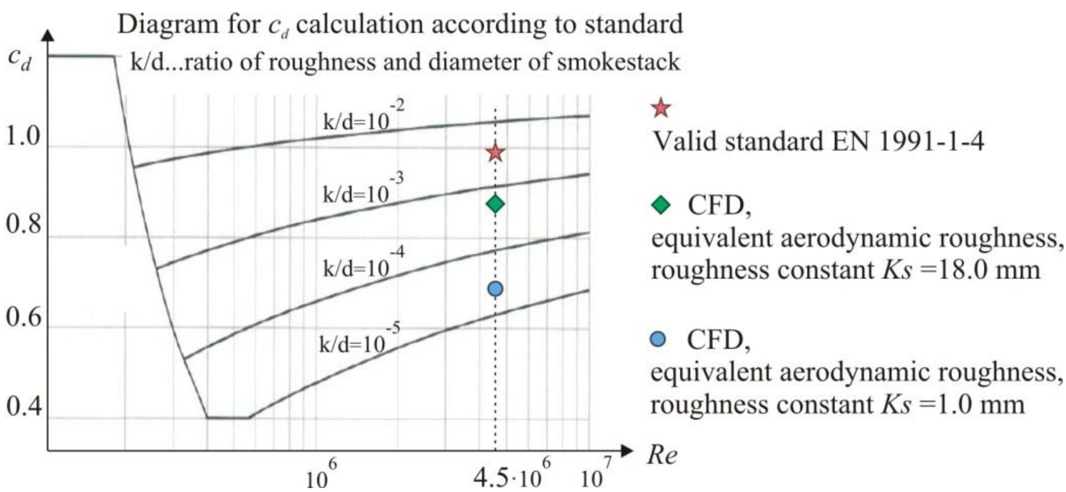

Fig.6: Drag coefficient values– modified scheme from EN 1991-1-4

4 CONCLUSION

The results of this work show that the drag coefficient comes out identically for both calculated geometrically similar sheets. The value of the drag coefficient determined by CFD codes using wall functions varies depending on the defined equivalent aerodynamic roughness. The value of c_d is about 13 % lower than that given in EN 1991-1-4 by calculation with an aerodynamic roughness corresponding to the actual wave height (roughness constant $K_S = 18.0$ mm). In the calculation of the equivalent aerodynamic roughness ($K_S = 1.0$ mm), which was defined on the basis of the results of a flow in a pipe with a wave height of 18 mm, the value of the coefficient is even 33 % lower. Results obtained using CFD codes are for information only. The conclusions can be stated with certainty that wind effects on a smokestack with a shaped sheet covering are in valid standard overvalued.

For confirmation, the results of numerical simulations should be verified by using either a physical experiment or a detailed numerical calculation, which allows the direct simulation of the real geometry of a covered smokestack. Such task, however, would require a powerful supercomputer, for example, that available in the National supercomputing center in Ostrava (http://www.it4i.cz/). This problem will be the subject of further research of the authors.

ACKNOWLEDGMENT

This contribution is the result of the research supported by the conceptual development of science, research and innovation for 2016 allocated to the VSB-Technical University of Ostrava by the Ministry of Education, Youth and Sports of the Czech Republic and also by specific grant Support for science and research in the Moravian-Silesian Region in 2014 given by Moravian-Silesian Region.

REFERENCES

[1] MERRICK, R. BITSUAMLAK, G. *Control of flow around a circular cylinder by the use of surface roughness: A computational and experimental approach*. In Proceeding of the 4th International Conference Advances on Wind and Structures (AWAS08). Jeju, Korea, 2008.

[2] DOBES, J. & KOZUBKOVA, M. The influence of numerical models on determining the drag coefficient. In *Proceeding of the International Conference on Experimental Fluid Mechanics*. Kutná Hora, Czech Republic, 2013. EPJ Web of Conferences.

[3] MICHALCOVÁ, V., KUZNĚTSOV, S. & POSPÍŠIL, S. Numerical modelling of air flow attributes in a Contractions chamber. *Transactions of the VŠB - Technical University of Ostrava*: *Construction Series* [online]. Warsaw, Poland: Versita, 2014, Vol. 14, Issue 2, pp. 11-16 (6 pp). ISSN 1804-4824 (Online); ISSN 1213-1962 (Print). DOI: 10.2478/tvsb-2014-0026.

[4] LAUSOVÁ, L., MATEČKOVÁ, P. & SKOTNICOVÁ, I. Experimental and numerical analysis of sSteel frame structure exposed to high temperature. *Transactions of the VŠB - Technical University of Ostrava*: *Construction Series* [online]. Warsaw, Poland: Versita, 2014, Vol. 14, Issue 2, pp. 64-68 (5 pp). ISSN 1804-4824 (Online); ISSN 1213-1962 (Print). DOI: 10.2478/tvsb-2014-0022.

[5] LAKEHAL, D. Computation of turbulent shear flows over rough-walled circular cylinders. *Journal of Wind Engineering and Industrial Aerodynamics*. 1999, LXX. Nr. 1-2, pp. 47-68. ISSN 0167-6105.

[6] XU, CH., CHEN, L., LU, X. Large-Eddy and Detached-Eddy Simulations of the separated flow around a circular cylinder. *Journal of Hydrodynamic*. 2007, IXX, Nr. 5,+ pp. 559-563. ISSN 1001-6058.

[7] SQUIRES, K. D., KRISHNAN, V. & FORSYTHE, J. R. Prediction of the flow a circular cylinder at high Reynolds number using detached-eddy simulation. *Journal of Wind Engineering and Industrial Aerodynamics*. 2008, XCVI, pp. 1528-1536. ISSN 0167-6105.

[8] NISHINO, T., ROBERTS, G.T., ZHANG, X. Unsteady RANS and detached-eddy simulations of flow around a circular cylinder in ground effect. *Journal of Fluids and Structures*. 2008, XXIV, Nr. 1, pp. 18-33, ISSN 0889-9746.

[9] KOTRASOVA; K., GRAJCIAR, I., KORMANIKOVA, E. Dynamic Time-History Response of Cylindrical Tank Considering Fluid - Structure Interaction Due to Earthquake. *Applied Mechanics and Materials*. Vol. 617, pp 66-69, 2014.

[10] KRALIK, J., KRALIK, J., JR. Failure Probability of NPP Communication Bridge under the Extreme Loads. *Applied Mechanics and Materials*. Vol. 617, pp 81-85, 2014.

[11] CATALANO, P., WANG, M. & IACCARINO, G. Numerical simulation of the flow around a circular cylinder at high Reynolds numbers. *Journal of Heat and Fluid Flow*. 2003, XXIV, pp. 463-469. ISSN 0142-727X.

[12] BREUER, M. A challenging test case for large eddy simulation: high Reynolds number circular cylinder flow. *Journal of Heat and Fluid Flow*. 2000, XXI. Nr. 1, pp. 648-654. ISSN 0142-727X.

[13] WANG, M., CATALANO, P., IACCARINO, G. Prediction of high Reynolds number flow over a circular cylinder using LES with wall modelling. *Cenrter for Turbulence Research*. pp. 45-50, Stanford University, 2001.

[14] ROSHKO, A. Experiments on the flow past a circular cylinder at very high Reynolds number. *Journal of Fluid Mechanics*. 2006, X. Nr. 1, pp. 345-356. ISSN 0022-1120.

[15] HUBOVA, O., KONECNA, L., LOBOTKA, P. Influence of Walls and Ceiling on a Wind Flow in BLWT Tunnel. *Applied Mechanics and Materials*. Vol. 617, pp 257-262, 2014.

[16] SHUR, M. L. Unsteady A hybrid RANS-LES approach with delayed-DES and wall-modelled LES capabilities. *Journal of Heat and Fluid Flow*. 2008, XXIX, pp.1638-1649. ISSN 0142-727X.

[17] KOLAR, V., VINOPAL, S. *Hydraulics of industrial valves (Hydraulika prumyslovych armatur)*. Guide for practical calculations. Praha: SNTL, 1963. 650 pp.