Computers and Structures xxx (2017) xxx–xxx

Contents lists available at ScienceDirect

Computers and Structures

journal homepage: www.elsevier.com/locate/compstruc

Numerical approach to determination of equivalent aerodynamic roughness of Industrial chimneys

V. Michalcova *, L. Lausova

Faculty of Civil Engineering, VŠB-Technical University of Ostrava, Ostrava-Poruba, Czech Republic

ARTICLE INFO

Article history:
Accepted 22 March 2017
Available online xxxx

Keywords:
CFD
Detached-eddy simulation (DES)
Circular cylinder
High Reynolds number
Boundary layer
Wall function
Aerodynamic roughness
Drag coefficient
Lift coefficient
Strouhal number

ABSTRACT

In building practices, smokestacks are commonly sheathed with smooth metal plates. The wind load acting on such sheathed smokestacks can be calculated according to applicable standards. However, requirements for sheathing smokestacks with corrugated metal have been recently increasing due to dilatation of the metal plate. For these cases, the calculation standards do not define parameters for determining the sheathing roughness that affects the load size. The calculation standards take into account only air flow around a cylinder with a coarse (rough) surface. In the case of a greater unevenness of sheathing of the flown around body, the standard takes into account only the roughness height, in this case, the metal corrugation height regardless of its type and shape. This fact may result in a large increase in frontal wind resistance.

One of the options to determine the load acting on the flown around smokestack sheathed with metal other than conventional smooth metal is a numerical solution. Creation of a corrugated sheathing mesh is demanding on the number of cells in the calculation area, and therefore, this approach is unrealistic for solution on desktop PCs at the present. The purpose of this numerical study is to determine an appropriate substitute, equivalent aerodynamic roughness, and thus the possibility of modelling in a simplified calculation area using the “wall function”. A properly determined equivalent aerodynamic roughness contributes to the correct definition of the drag coefficient value that defines the size of the wind load acting on the flown around object (in Fluent drag coefficient).

The presented thesis simulates airflow around a realistic smokestack of a circular section for two different types of sheathing - corrugated and trapezoidal plates. The task is dealt with using the finite volume method using CFD codes in Ansys Fluent software. The numerical solution results are evaluated and compared with standard regulations.

1. Introduction

The paper is based upon Michalcova and Lausova [1], where the issue was focused on determining the sheathing roughness of high smokestacks. The paper [1] presented the option to determine the equivalent aerodynamic roughness required for the solution using the wall function [2].

The current paper is expanded with the thought of determining the equivalent aerodynamic roughness of the flown around smokestack by two possible alternatives so that it complies with the requirements of the calculation standard. For the purposes of evaluating the results of both alternatives, the flow pattern in the smokestack wake is monitored. We monitor the lift coefficient so as to be able to define the Strouhal number [3], defining the eddy detaching process [4]. The purpose of this study is to show the approach to determining sheathing roughness of high smokestacks that has a big impact on the load calculation.

Smokestack sheathing with conventional smooth non-shaped metal is a standard solution, associated with certain complications. This is caused by material dilatation that is a problem mainly in large smokestacks (Fig. 1). Undesirable phenomena are also supported by a combination of different materials of the supporting shell and external cladding (steel × aluminum). This is the reason for frequently imposing the requirement for the smokestack sheathing with shaped metal. The problem occurs when dimensioning such a structure, in particular, when determining the wind drag coefficient.

Calculation according to the applicable standard EN 1991-1-4 only takes into account the airflow around the cylinder with a rough surface (small height of surface unevenness). The most adverse cases include raw wood, rust or brick walls. The standard

* Corresponding author.
E-mail address: vladimira.michalcova@vsb.cz (V. Michalcova).

http://dx.doi.org/10.1016/j.compstruc.2017.03.013
0045-7949/

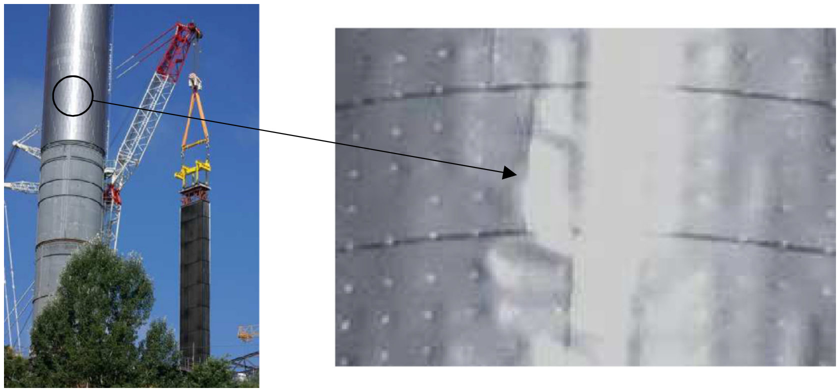

Fig. 1. Deformation of smokestack sheathing.

does not stipulate what equivalent surface roughness is to be allocated to a cylinder with a smooth corrugation type surface. It only recommends determining the equivalent surface roughness close to the height of the unevenness (i.e. the metal corrugation height) regardless of its shape. In many cases, it results in a high increase in the drag coefficient, referred to in the standard as the force coefficient $c_{f,0}$. The increase is often twice as much.

The purpose of this paper is to use CFD (Computational Fluid Dynamics) codes in the Ansys Fluent software to find options to define the substitute - equivalent aerodynamic roughness value and subsequently the drag coefficient value c_d [5,6], determining the size of wind load acting on the structure. It is equivalent to the force coefficient $c_{f,0}$ in the standard calculation.

2. Airflow around a cylinder at high *Re* numbers

The task of airflow around a cylinder with a flow rate at high Reynolds numbers is a complex phenomenon, the solution of which is dealt with by many international departments, whether in experimental [7] or numerical [8] research.

The drag coefficient c_d value depends on the Reynolds number and is highly susceptible to the roughness of the flow around the surface [9,10]. The higher the roughness, the lower the *Re* of the transition area and the higher the minimum c_d value (Fig. 2). This issue also finds its use in other areas, such as dealing with the dynamics of a flown around cylinder [11], optimizing bridge structures [12] or preparing models of flown around objects [13].

This paper simulates airflow around a realistic smokestack of a circular section with a diameter of 3.36 m. Sheathing consists of a specific corrugated and trapezoidal metal. Both have the depth of 18 mm, as well as relatively close geometry. Basic wind speed is assumed $v = 20$ m/s. The air flow rate is thus defined with the Reynolds number around $Re = 4.5 \cdot 10^6$ and is well into the supercritical area [14,15]. Therefore, fully advanced turbulence can be assumed in the boundary layer around the smokestack wall [16,17]. The basic assumption for the objective numerical solution of wind load acting on the smokestack is the correct setting of calculations near the wall of the flown around body.

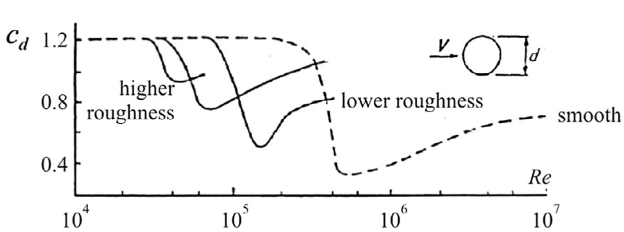

Fig. 2. c_d coefficient - *Re* number diagram.

3. Near-wall numerical wind flow modelling

The subject features change quickly near the wall, the transition of dynamics and scalar quantities assert themselves here significantly. Turbulence near the wall (in the viscous sub-layer and transition layer) is suppressed, however, turbulent kinetic energy production is intensive in the external part of the boundary layer. It is appropriate to cooperate with experimental research for the solution [18,19].

The near-wall flow can be modelled in two ways. The first uses the wall function [2], thanks to which it bypasses the area of the laminar sub-layer and transition layer. This is an area between the wall and the area of fully developed turbulent flow where there is molecular and turbulent viscosity. The solution using the wall function is highly susceptible not only to the correct creation of the mesh near the wall, but also to the correct definition of equivalent aerodynamic roughness [20] with which this paper deals.

The second method of solution consists in detailed near-wall modelling including the viscous sub-layer in relation to mesh fineness [21]. For calculation with near-wall modelling with the real geometry of smokestack sheathing, the requirement for a number of cells in the calculation area is currently unrealistic for solution on desktop PCs. It is necessary to use a high-capacity supercomputer for detailed numerical calculation that enables direct simulation of real geometry [22].

Near-wall flow modelling of a profiled smokestack sheathing thus leads to use of the wall function that needs to be defined correctly.

3.1. Wall function in Ansys Fluent

This is modelling of the airflow around a non-profiled cylinder with a significantly lower number of cells. The influence of the actual sheathing shape substitutes for the use of the wall function that constitutes a set of semi-empirical relations and functions applied in the first cell from the wall. The wall function includes the law of the wall for mean velocity and relations for turbulent near-wall quantities [21].

The wall function is defined by the modified logarithmic law:

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

where

B is an additive constant describing the roughness function [–],
κ Von Kármán constant [–] and